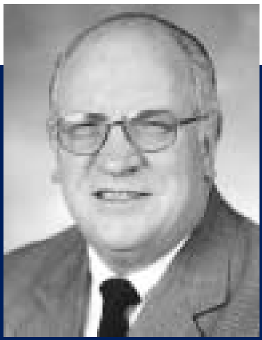
CHAPTER 12 Milling Cutters and Operations



George Schneider, Jr. CMfgE
Professor Emeritus
Engineering Technology
Lawrence Technological University

Former Chairman
Detroit Chapter ONE
Society of Manufacturing Engineers

Former President
International Excutive Board
Society of Carbide & Tool Engineers

Lawrence Tech.- www.ltu.edu

Prentice Hall- www.prenhall.com

12.1 Introduction

The two basic cutting tool types used in the metal working industry are of the single point and multi-point design, although they may differ in appearance and in their methods of application. Fundamentally, they are similar in that the action of metal cutting is the same regardless of the type of operation. By grouping a number of single point tools in a circular holder, the familiar milling cutter is created.

Milling is a process of generating machined surfaces by progressively removing a predetermined amount of material or stock from the workpiece witch is advanced at a relatively slow rate of movement or feed to a milling cutter rotating at a comparatively high speed. The characteristic feature of the milling process is that each milling cutter tooth removes its share of the stock in the form of small individual chips. A typical face milling operation is shown in Figure 12.1.

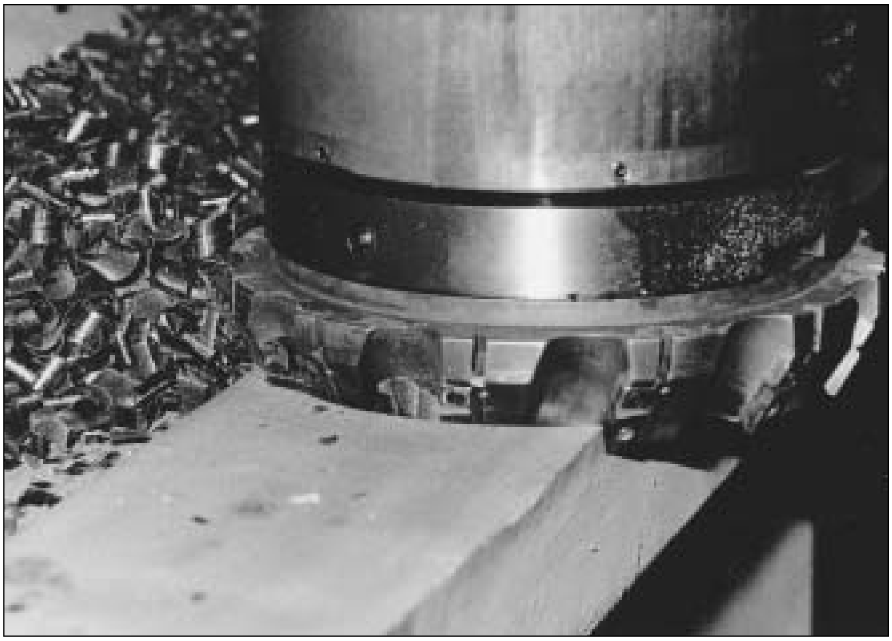


FIGURE 12.1: A typical milling operation; the on-edge insert design is being used. (Courtesy Ingersoll Cutting Tool Co.)

12.2 Types of Milling Cutters

The variety of milling cutters available for all types of milling machines helps make milling a very versatile machining process. Cutters are made in a large range of sizes and of several different cutting tool materials. Milling cutters are made from High Speed Steel (HSS), others are carbide tipped and many are replaceable or indexable inserts. The three basic milling operations are shown in Figure 12.2. Peripheral and end milling cutters will be discussed below. Face

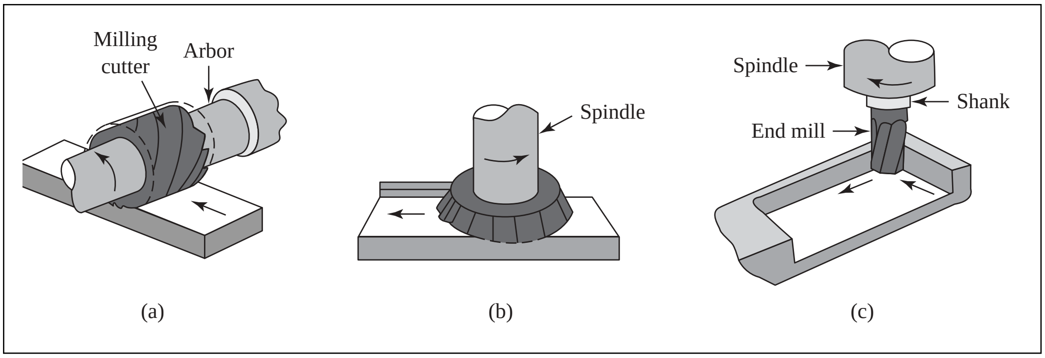


FIGURE 12.2: The three basic milling operations: (a) milling, (b) face milling, (c) end milling

FIGURE 12.3: High-speed steel (HSS) shell end milling cutter. (Courtesy Morse Cutting Tools)

milling cutters are usually indexable and will be discussed later in this chapter.

A high speed steel (HSS) shell end milling cutter is shown in Figure 12.3 and other common HSS cutters are shown in Figure 12.4 and briefly described below:

12.2.1 Periphery Milling Cutters

Periphery milling cutters are usually arbor mounted to perform various operations.

Light Duty Plain Mill: This cutter is a general purpose cutter for peripheral milling operations. Narrow cutters have straight teeth, while wide ones have helical teeth (Fig. 12.4c).

Heavy Duty Plain Mill: A heavy duty plain mill is similar to the light duty mill except that it is used for higher rates of metal removal. To aid it in this function, the teeth are more widely spaced and the helix angle is increased to about 45 degrees.

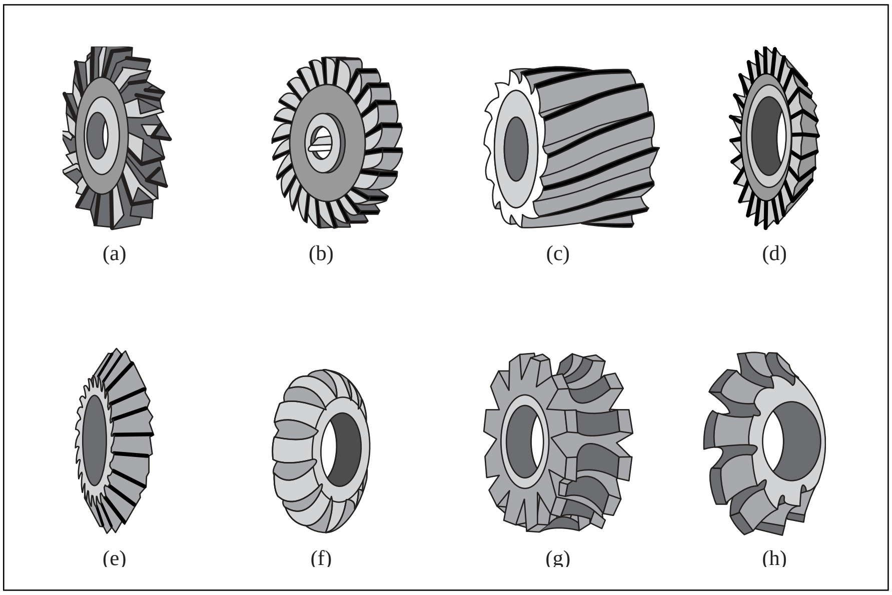


FIGURE 12.4: Common HSS milling cutters: (a) staggered-tooth cutter, (b) side milling cutter, (c) plain milling cutter, (d) single-angle milling cutter, (e) double-angle milling cutter, (f) convex milling cutter, (g) concave milling cutter, (h) corner rounded milling cutter.

Side Milling Cutter: The side milling cutter has a cutting edge on the sides as well as on the periphery. This allows the cutter to mill slots (Fig. 12.4b).

Half-Side Milling Cutter: This tool is the same as the one previously described except that cutting edges are provided on a single side. It is used for milling shoulders. Two cutters of this type are often mounted on a single arbor for straddle milling.

Stagger-tooth Side Mill: This cutter is the same as the side milling cutter except that the teeth are staggered so that every other tooth cuts on a given side of the slot. This allows deep, heavy-duty cuts to be taken (12.4a).

Angle Cutters: On angle cutters, the peripheral cutting edges lie on a cone rather than on a cylinder. A single or double angle may be provided (Fig. 12.4d and Fig. 12.4e).

Shell End Mill: The shell end mill has peripheral cutting edges plus face cutting edges on one end. It has a hole through it for a bolt to secure it to the spindle (Fig. 12.3).

Form Mill: A form mill is a peripheral cutter whose edge is shaped to produce a special configuration on the surface. One example of his class of tool is the gear tooth cutter. The exact contour of the cutting edge of a form mill is reproduced on the surface of the workpiece (Fig.12.4f, Fig.12.4g, and Fig.12.4h).

12.2.2 End Milling Cutters

End mills can be used on vertical and horizontal milling machines for a variety of facing, slotting, and profiling operations. Solid end mills are made from high speed steel or sintered carbide. Other types, such as shell end mills and fly cutters, consist of cutting tools that are bolted or otherwise fastened to adapters.

Solid End Mills: Solid end mills have two, three, four, or more flutes and cutting edges on the end and the periphery. Two flute end mills can be fed directly along their longitudinal axis into solid material because the cutting faces on the end meet. Three

FIGURE 12.5a: Various single- and double-ended HSS end mills. (Courtesy The Weldon Tool Co.)

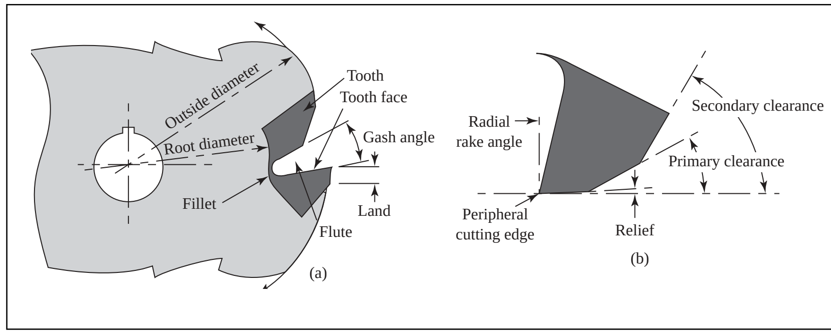
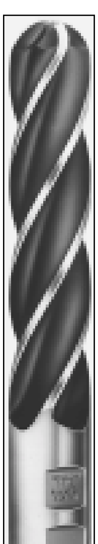


FIGURE 12.7: Milling cutter configuration: (a) plain milling cutter nomenclature; (b) plain milling cutter tooth geometry.

FIGURE 12.5b: Various tapered HSS end mills. (Courtesy The Weldon Tool Co.)



(a) (b)

FIGURE 12.6: (a) Ball-nose end-milling cutters are available in diameter ranging from 1/32 to 2 ½ inches. (Courtesy The Weldon Tool Co.) (b) HSS dovetail cutters can be used on both vertical and horizontal milling machines. (Courtesy Morse Cutting Tools)

and four fluted cutters with one end cutting edge that extends past the center of the cutter can also be fed directly into solid material.

Solid end mills are double or single ended, with straight or tapered shanks. The end mill can be of the stub type, with short cutting flutes, or of the extra long type for reaching into deep cavities. On end mills designed for effective cutting of aluminum, the helix angle is increased for improved shearing action and chip removal, and the flutes may be polished. Various single and double-ended end mills are shown in Figure 12.5a. Various tapered end mills are shown in Figure 12.5b.

Special End Mills: Ball end mills (Fig. 12.6a) are available in diameters ranging from 1/32 to 2 1/2 inches in single and double ended types. Single purpose end mills such as Woodruff key-seat cutters, corner rounding cutters, and dovetail cutters (Fig.12.6b) are used on both vertical and horizontal milling machines. They are usually made of high speed steel and may have straight or tapered shanks.

12.3 Milling Cutter Nomenclature

As far as metal cutting action is concerned, the pertinent angles on the tooth are those that define the configuration of the cutting edge, the orientation of the tooth face, and the relief to prevent rubbing on the land.

The terms defined below and illustrated in Figures 12.7a and 12.7b are important and fundamental to milling cutter configuration.

Outside Diameter: The outside diameter of a milling cutter is the diameter of a circle passing through the peripheral cutting edges. It is the dimension used in conjunction with the spindle speed to find the cutting speed (SFPM).

Root Diameter: This diameter is measured on a circle passing through the bottom of the fillets of the teeth.

Tooth: The tooth is the part of the cutter starting at the body and ending with the peripheral cutting edge. Replaceable teeth are also called inserts.

Tooth Face: The tooth face is the surface of the tooth between the fillet and the cutting edge, where the chip slides during its formation.

Land: The area behind the cutting edge on the tooth that is relieved to avoid interference is called the land.

Flute: The flute is the space provided for chip flow between the teeth.

Gash Angle: The gash angle is measured between the tooth face and the back of the tooth immediately ahead.

Fillet: The fillet is the radius at the bottom of the flute, provided to allow chip flow and chip curling.

The terms defined above apply primarily to milling cutters, particularly to plain milling cutters. In defining the configuration of the teeth on the cutter, the following terms are important.

Peripheral Cutting Edge: The cutting edge aligned principally in the direction of the cutter axis is called the peripheral cutting edge. In peripheral milling, it is this edge that removes the metal.

FIGURE 12.8: A variety of indexable milling cutters. (Courtesy Ingersoll Cutting Tool Co.)

(a)

(b)

FIGURE 12.10: (a) Face milling cutter with wedge clamped indexable inserts. (Courtesy Iscar Metals, Inc.) (b) Face milling cutters with indexable inserts and wedge clamped milling cartridges. (Courtesy Greenleaf Corp.)

Face Cutting Edge: The face cutting edge is the metal removing edge aligned primarily in a radial direction. In side milling and face milling, this edge actually forms the new surface, although the peripheral cutting edge may still be removing most of the metal. It corresponds to the end cutting edge on single point tools.

Relief Angle: This angle is measured between the land and a tangent to the cutting edge at the periphery.

Clearance Angle: The clearance angle is provided to make room for chips, thus forming the flute. Normally two clearance angles are provided to maintain the strength of the tooth and still provide sufficient chip space.

Radial Rake Angle: The radial rake angle is the angle between the tooth face and a cutter radius, measured in a plane normal to the cutter axis.

Axial Rake Angle: The axial rake angle is measured between the peripheral cutting edge and the axis of the cutter, when looking radially at the point of intersection.

Blade Setting Angle: When a slot is provided in the cutter body for a blade, the angle between the base of the slot and the cutter axis is called the blade setting angle.

12.4 Indexable Milling Cutters

The three basic types of milling operations were introduced earlier. Figure 12.8 shows a variety of indexable milling cutters used in all three of the basic types of milling operations (Fig. 12.2).

There are a variety of clamping systems for indexable inserts in milling cutter bodies. The examples shown cover the most popular methods now in use:

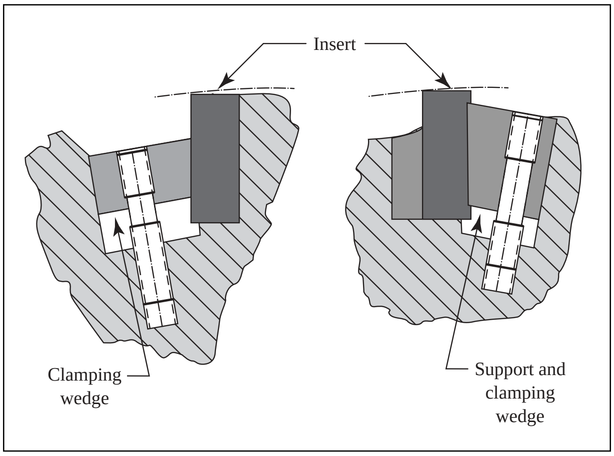


FIGURE 12.9: Two methods of wedge clamping indexable milling cutter inserts.

12.4.1 Wedge Clamping

Milling inserts have been clamped using wedges for many years in the cutting tool industry. This principle is generally applied in one of the following ways: either the wedge is designed and oriented to support the insert as it is clamped, or the wedge clamps on the cutting face of the insert, forcing the insert against the milling body. When the wedge is used to support the insert, the wedge must absorb all of the force generated during the cut. This is why wedge clamping on the cutting face of the insert is preferred, since this method transfers the loads generated by the cut through the insert and into the cutter body. Both of the wedges clamping methods are shown in Figure 12.9.

The wedge clamp system however, has two distinct disadvantages. First, the wedge covers almost half of the insert cutting face, thus obstructing normal chip flow while producing premature cutter body wear, and secondly, high clamping forces causing clamping element and cutter body deformation can and often will result. The excessive clamping forces can cause enough cutter body distortion that in some cases when loading inserts into a milling body, the last insert slot will have narrowed to a point where the last insert will not fit into the body. When this occurs, several of the other inserts already loaded in the milling cutter are removed an reset. Wedge clamping can be used to clamp individual inserts (Fig. 12.10a) or indexable and replaceable milling cutter cartridges as shown in Figure 25.10b.

12.4.2 Screw Clamping

This method of clamping is used in conjunction with an insert that has a pressed countersink or counterbore. A torque screw is often used to eccentrically mount and force the insert against the insert pocket walls. This clamping action is a result of either offsetting the centerline of the screw toward the back walls of the insert

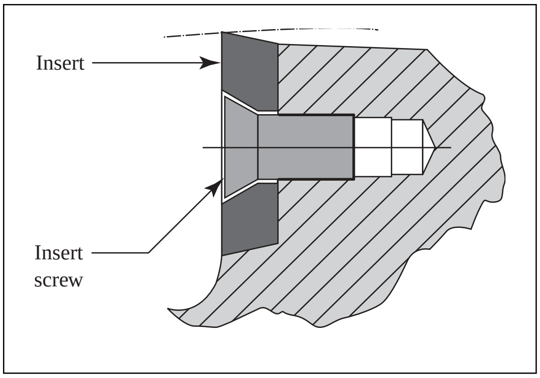


FIGURE 12.11: Screw clamping method for indexable inserts.

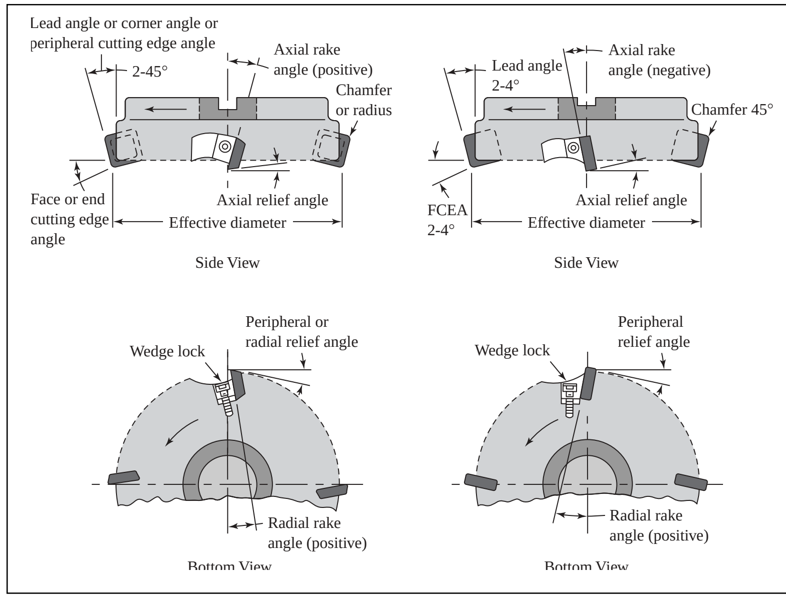


FIGURE 12.13: Positive-rake and negative-rake face milling cutter nomenclature.

pocket, or by drilling and tapping the mounting hole at a slight angle, thereby bending the screw to attain the same type of clamping action.

The Screw clamping method for indexable inserts is shown in Figure 12.11.

Screw clamping is excellent for small diameter end mills where space is at a premium. It also provides an open unhampered path for chips to flow free of wedges or any other obstructive hardware. Screw clamping produces lower clamping forces than those attained with the wedge clamping system. However, when the cutting edge temperature rises significantly, the insert frequently expands and causes an undesirable retightening effect, increasing the torque required to unlock the insert screw. The screw clamping method can be used on indexable ball milling cutters (Fig. 12.12a) or on indexable insert slotting and face milling cutters as shown in Figure 12.12b.

12.5 Milling Cutter Geometry

There are three industry standard milling cutter geometries: double negative, double positive, and positive/negative. Each cutter geometry type has certain advantages and disadvantages that must be considered when selecting the right milling cutter for the job. Positive rake and negative rake milling cutter geometries are shown in Figure 12.13.

Double Negative Geometry: A double negative milling cutter uses only negative inserts held in a negative pocket. This provides cutting edge strength for roughing and severe interrupted cuts. When choosing a cutter geometry it is important to remember that a negative insert tends to push the cutter away, exerting considerable force against the workpiece. This could be a problem when machining flimsy or lightly held workpieces, or when using light machines. However, this tendency to push the work down, or push the cutter away from the workpiece may be beneficial in some cases because the force tends to 'load' the system, which often reduces chatter.

Double Positive Geometry: Double positive cutters use positive inserts held in positive pockets. This is to provide the proper clearance for cutting. Double positive cutter geometry provides for low force cutting, but the inserts contact the workpiece at their weakest point, the cutting edge. In positive rake milling, the cutting forces tend to lift the workpiece or pull the cutter into the work. The greatest advantage of double posi-

(a)

(b)

FIGURE 12.12: (a) Indexable-insert ball-nosed milling cutters using the screw clamping method. (Courtesy Ingersoll Cutting Tool Co.) (b) Slotting cutters and face milling with screw-on-type indexable inserts. (Courtesy Duramet Corp.)

tive milling is free cutting. Less force is exerted against the workpiece, so less power is required. This can be especially helpful with machining materials that tend to work harden.

Positive / Negative Geometry: Positive/negative cutter geometry combines positive inserts held in negative pockets. This provides a positive axial rake and a negative radial rake and as with double positive inserts, this provides the proper clearance for cutting. In the case of positive/negative cutters, the workpiece is contacted away from the cutting edge in the radial direction and on the cutting edge in the axial direction. The positive/negative cutter can be considered a low force cutter because it uses a free cutting positive insert. On the other hand, the positive/negative cutter provides contact away from the cutting edge in the radial direction, the feed direction of a face mill.

In positive/negative milling, some of the advantages of both positive and negative milling are available. Positive/negative milling combines the free cutting or shearing away of the chip of a positive cutter with some of the edge strength of a negative cutter.

Lead Angle: The lead angle (Fig. 12.14) is the angle between the insert and the axis of the cutter. Several factors must be considered to determine which lead angle is best for a specific operation. First, the lead angle must be small enough to cover the depth of cut. The greater the lead angle, the less the depth of cut that can be taken for a given size insert. In addition, the part being machined may require a small lead angle in order to clear a portion or form a certain shape on the part. As the lead angle increases, the forces change toward the direction of the workpiece. This could cause deflections when machining thin sections of the part.

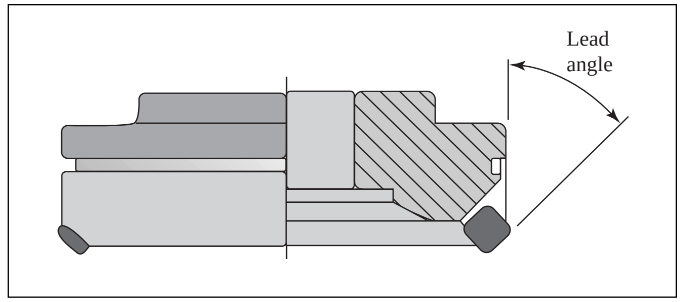


FIGURE 12.14: Drawing of a positive lead angle on an indexable-insert face milling cutter.

The lead angle also determines the thickness of the chip. The greater the lead angle for the same feed rate or chip load per tooth, the thinner the chip becomes. As in single point tooling, the depth of cut is distributed over a longer surface of contact. Therefore, lead angle cutters are recommended when maximum material removal is the objective. Thinning the chip allows the feed rate to be increased or maximized.

Lead angles can range from zero to 85 degrees. The most common lead angles available on standard cutters are 0, 15, 30 and 45 degrees. Lead angles larger than 45 degrees are usually considered special, and are used for very shallow cuts for fine finishing, or for cutting very hard work materials.

Milling cutters with large lead angles also have greater heat dissipating capacity. Extremely high temperatures are generated at the insert cutting edge while the insert is in the cut. Carbide, as well as other tool materials, often softens when heated, and when a cutting edge is softened it will wear away more easily. However, if more of the tool can be employed in the cut, as in the case of larger lead angles, the tool's heat dissipating capacity will be improved which, in turn, improves tool life. In addition, as lead angle is increased, axial force is increased and radial force is reduced, an important factor in controlling chatter.

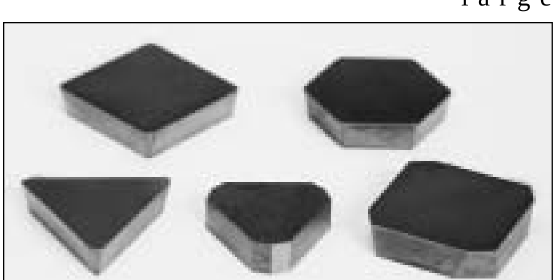
The use of large lead angle cutters is especially beneficial when machining materials with scaly or work hardened surfaces. With a large lead angle, the surface is spread over a larger area of the cutting edge. This reduces the detrimental effect on the inserts, extending tool life. Large lead angles will also reduce burring and breakout at the workpiece edge.

The most obvious limitation on lead angle cutters is part configuration. If a square shoulder must be machined on a part, a zero degree lead angle is required. It is impossible to produce a zero degree lead angle milling cutter with square inserts because of the need to provide face clearance. Often a near square shoulder is permissible. In this case a three degree lead angle cutter may be used.

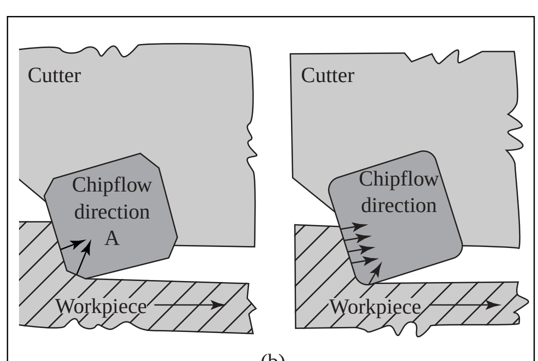
12.5.1 Milling Insert Corner Geometry

Indexable insert shape and size were discussed in Chapter 2. Selecting the proper corner geometry is probably the most complex element of insert selection. A wide variety of corner styles are available. The corner style chosen will have a major effect on surface finish and insert cost. Figure 12.15a shows various sizes and shapes of indexable milling cutter inserts.

Nose Radius: An insert with a nose radius is generally less expensive than a similar insert with any other corner geometry. A nose radius is also the strongest possible corner geometry because it has no sharp corners where two flats come together, as in the case of a chamfered corner. For these two



(a)



(b)

FIGURE 12.15: (a) Various sizes and shapes of indexable milling cutter inserts. (Courtesy American National Carbide Co.) (b) indexable milling cutter insert chip flow directions are shown.

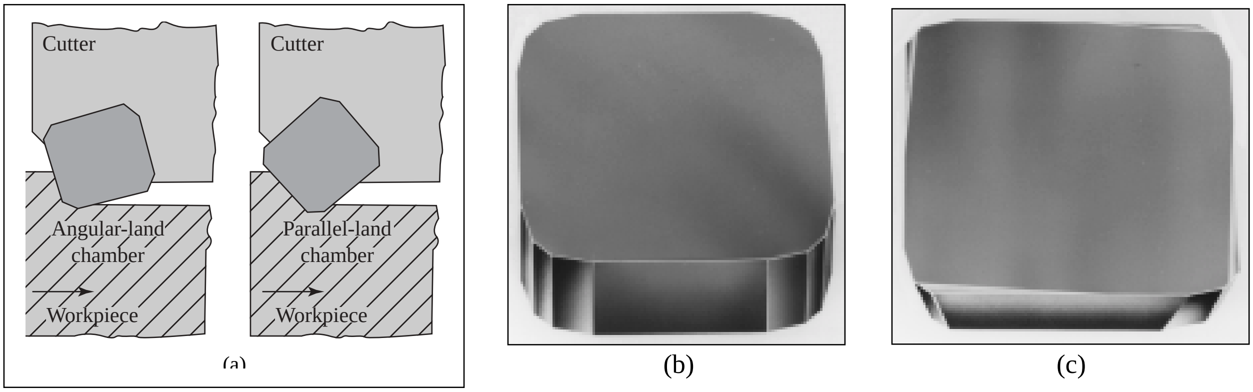


FIGURE 12.16: (a) indexable milling cutter inserts with angular-land chamfer and parallel-land chamfer. (b and c) Two precision ground indexable milling cutter inserts. (Courtesy Iscar Metals, Inc.)

reasons alone, a nose radius insert should be the first choice for any application where it can be used.

Inserts with nose radii can offer tool life improvement when they are used in 0 to 15 degree lead angle cutters, as shown in Figure 12.15b. When a chamfer is used, as in the left drawing, the section of the chip formed above and below point A, will converge at point A, generating a large amount of heat at that point, which will promote faster than normal tool wear. When a radius insert is used, as shown in the right drawing, the chip is still compressed, but the heat is spread more evenly along the cutting edge, resulting in longer tool life.

The major disadvantage of an insert with a nose radius is that the surface finish it produces is generally not as good as other common corner geometries. For this reason, inserts with nose radii are generally limited to roughing applications and applications where a sweep wiper insert is used for the surface. A sweep wiper is an insert with a very wide flat edge or a very large radiused edge that appears to be flat. There is usually only one wiper blade used in a cutter and this blade gets its name from its sweeping action that blends the workpiece surface to a very smooth finish.

Inserts with nose radii are not available on many double positive and positive/negative cutters because the clearance required under the nose radius is different from that needed under the edge. This clearance difference would require expensive grinding procedures that would more than offset the other advantages of nose radius inserts.

Chamfer: There are two basic ways in which inserts with a corner chamfer can be applied. Depending both on the chamfer angle and the lead angle of the cutter body in which the insert is used, the land of the chamfer will be either parallel or angular (tilted) to the direction of feed, as shown in Figure 12.16a.

Inserts that are applied with the chamfer angular to the direction of feed normally have only a single chamfer. These inserts are generally not as strong and the cost is usually higher than inserts that have a large nose radius. Angular-land chamfer inserts are frequently used for general purpose machining with double negative cutters.

Inserts designed to be used with the chamfer parallel to the direction of feed may have a single chamfer, a single chamfer and corner break, a double chamfer, or a double chamfer and corner break. The larger lands are referred to as primary facets and the smaller lands as secondary facets. The cost of chamfers, in relation to other types of corner geometries, depends upon the number of facets. A single facet insert is the least expensive, while multiple facet inserts cost more because of the additional grinding expense. Figure 12.16b shows two precision ground indexable milling cutter inserts. A face milling cutter with six square precision ground indexable milling cutter inserts was shown in Figure 12.10a.

The greatest advantage of using inserts with the land parallel to the direction of feed is that, when used correctly, they generate an excellent surface finish. When the land width is greater than the advance per revolution, one insert forms the surface. This means that an excellent surface finish normally will be produced regardless of the insert face runout. Parallel-land inserts also make excellent roughing and general purpose inserts for positive/negative and double positive cutters. When a parallel land chamfer insert is used for roughing, the land width should be as small as possible to reduce friction.

Sweep Wipers: Sweep wipers are unique in both appearance and application. These inserts have only one or two very long wiping lands. A single sweep wiper is used in a cutter body filled with other inserts (usually roughing inserts) and is set approximately 0.003 to 0.005 inches higher than the other inserts, so that the sweep wiper alone forms the finished surface.

The finish obtained with a sweep wiper is even better than the excellent finish attained with a parallel land chamfer insert. In addition, since the edge of the sweep wiper insert is exceptionally long, a greater advance per revolution may be used. The sweep wiper also offers the same easy set-up as the parallel-land insert.

Sweep wiper inserts are available with both flat and crowned wiping surfaces. The crowned cutting edge is ground to a very large radius, usually from three to ten inches. The crowned cutting edges eliminate the possibility of saw-tooth profiles being produced on the machined surface because the land is not exactly parallel to the direction of feed, a condition normally caused by spindle tilt. On the other hand, sweep wipers with flat cutting edges produce a somewhat better finish if the land is perfectly aligned with the direction of feed.

12.6 Basic Milling Operations

Before any milling job is attempted, several decisions must be made. In addition to selecting the best means of holding the work and the most appropriate cutters to be used, the cutting speed and feed rate must be established to provide good balance between rapid metal removal and long tool life.

Proper determination of a cutting speed and feed rate can be made only when the following six factors are known:

- Type of material to be machined
- Rigidity of the set-up
- Physical strength of the cutter
- Cutting tool material
- Power available at the spindle
- Type of finish desired

Several of these factors affect cutting speed only, and some affect both cutting speed and the feed rate. The tables in reference handbooks provide approximate figures that can be used as starting points. After the cutting speed is chosen, the spindle speed must be computed and the machine adjusted.

Cutting Speed: Cutting speed is defined as the distance in feet that is traveled by a point on the cutter periphery in one minute. Since a cutter's periphery is its circumference:

Circumference = Pi × d

in case of a cutter, the circumference is:

Cutter circumference = Pi/12 × d
= .262 × d

Since cutting speed is expressed in surface feet per minute (SFPM)

SFPM = Cutter circumference × RPM

by substituting for the cutter circumference, the cutting speed can be expressed as:

SFPM = .262 × d × RPM

The concept of cutting speed (SFPM) was introduced in Chapter 4 (Turning Tools and Operations) and explained again in Chapter 8 (Drills and Drilling Operations). It has again been reviewed here without giving additional examples. However, since milling is a multi-point operation, feed needs to be explained in more detail than in previous chapters.

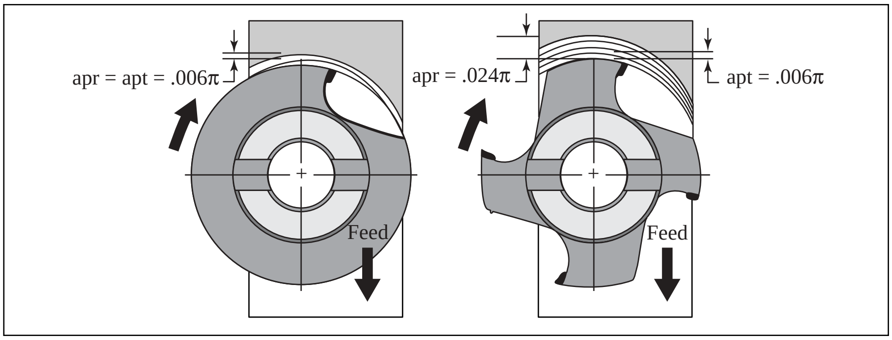


FIGURE 12.17: Drawing of a milling cutter showing the difference between advance per revolution (apr) and advance per tooth (apt).

Feed Rate: Once the cutting speed is established for a particular workpiece material, the appropriate feed rate must be selected. Feed rate is defined in metal cutting as the linear distance the tool moves at a constant rate relative to the workpiece in a specified amount of time. Feed rate is normally measured in units of inches per minute or IPM. In turning and drilling operations the feed rate is expressed in IPR or inches per revolution.

When establishing the feed rates for milling cutters, the goal is to attain the fastest feed per insert possible, to achieve an optimum level of productivity and tool life, consistent with efficient manufacturing practices. The ultimate feed rate is a function of the cutting edge strength and the rigidity of the workpiece, machine and fixturing. To calculate the appropriate feed rate for a specific milling application, the RPM, number of effective inserts (N) and feed per insert in inches (IPT or **apt**) should be supplied.

The milling cutter shown in Figure 12.17 on the left (one insert cutter) will advance .006 inches at the cutter centerline every time it rotates one full revolution. In this case, the cutter is said to have a feed per insert or an IPT (inches per tooth), **apt** (advance per tooth) and an **apr** (advance per revolution) of .006 inches. The same style of cutter with 4 inserts is shown in the right hand drawing. However, to maintain an equal load on each insert, the milling cutter will now advance .024 inches at he centerline every time it rotates one full revolution. The milling cutter on the right is said to have and IPT and **apt** of .006 inches, but and **apr** (advance per revolution) of .024 inches (.006 inch for each insert).

These concepts are used to determine the actual feed rate of a milling cutter in IPM (inches per minute) using one of the following formulas:

IPM = (IPT) × (N) × (RPM)
or
IPM = (apt) × (N) × (RPM)

where:
IPM = inches per minute
N = number of effective inserts
IPT = inches per tooth
apt = advance per tooth
RPM = revolutions per minute

*For Example: When milling automotive gray cast iron using a 4 inch diameter face mill with 8 inserts at 400 SFPM and 30.5 IPM, what **apr** and **apt** would this be?*

$$\text{RPM} = \frac{\text{SFPM}}{.262 \times d} = \frac{400}{.262 \times 4} = 382$$

$$\textbf{apr} = \frac{\text{IPM}}{\text{RPM}} = \frac{30.5}{382} = .080 \text{ in.}$$

$$\textbf{apt} = \frac{\textbf{apr}}{N} = \frac{.080}{8} = .010 \text{ in.}$$

Answer: = .080 in. **apr**
= .010 in. **apt**

When milling a 300M steel landing gear with a 6 inch diameter 45 degree lead face mill (containing 10 inserts) at 380 SFPM and a .006 inch advance per tooth, what feed rate should be run in IPM?

$$\text{RPM} = \frac{\text{SFPM}}{.262 \times d} = \frac{380}{.262 \times 6} = 242$$

$$\text{IPM} = \textbf{apt} \times N \times \text{RPM} = .006 \times 10 \times 242 = 14.5$$

Answer: = 14.5 IPM

The following basic list of formulas can be used to determine IPM, RPM, **apt, apr**, or N depending on what information is supplied for a specific milling application:

IPM = inches per minute
N = number of effective inserts
apr = inches of cutter advance every revolution
apt = inches of cutter advance for each effective insert every revolution
RPM = revolutions per minute

Find	**Given**	**Using**
IPM	**apr**, RPM	IPM = apr × RPM
IPM	RPM, N, **apt**	IPM = **apt** × N × RPM
apr	IPM, RPM	**apr** = IPM/RPM
RPM	IPM, **apr**	RPM = IPM/**apr**
RPM	IPM, N, **apt**	$\text{RPM} = \dfrac{\text{IPM}}{\text{N} \times \textbf{apt}}$
N	IPM, RPM, **apt**	$\text{N} = \dfrac{\text{IPM}}{\text{RPM} \times \textbf{apt}}$
apt	IPM, N, RPM	$\textbf{apt} = \dfrac{\text{IPM}}{\text{RPM} \times \text{N}}$

Note: In the formulas shown above IPT can be substituted for **apt** and IPR can be substituted for **apr**.

Horsepower Requirements: In metal cutting, the horsepower consumed is directly proportional to the volume (Q) of material machined per unit of time (cubic inches / minute). Metals have distinct unit power factors that indicate the average amount of horsepower required to remove one cubic inch of material in a minute. The power factor **(k*)** can be used either to determine the machine size in terms of horsepower required to make a specific machining pass or the feed rate that can be attained once a depth and width of cut are established on a particular part feature. To determine the metal removal rate (Q) use the following:

$$Q = \text{D.O.C.} \times \text{W.O.C} \times \text{IPM}$$

where:
D.O.C. = depth of cut in inches
W.O.C. = width of cut in inches
IPM = feed rate, in inches/minute

The average spindle horsepower required for machining metal workpieces is as follows:

$$\text{HP} = Q \times k^*$$

where:
HP = horsepower required at the machine spindle
Q = the metal removal rate in cubic inches/minute
k* = the unit power factor in HP/cubic inch/minute

***k factors are available from reference books**

For example: What feed should be selected to mill a 2 inch wide by .25 inch depth of cut on aircraft aluminum, utilizing all the available horsepower on a 20 HP machine using a 3 inch diameter face mill?

$$\text{HP} = Q \times k^*$$

k* = .25 H.P./in.^{3}/min. for aluminum

The maximum possible metal removal rate (Q), for a 20 H.P. machine running an aluminum part is:

$$Q = \frac{\text{HP}}{k} = \frac{20}{.25} = 80 \text{ in}^3\text{/min.}$$

Answer: Q = 80 in.^{3}/min.

To remove 80 in^3/min., what feed rate will be needed?

Q = (D.O.C.) × (W.O.C.) × IPM

$$\text{IPM} = \frac{Q}{(\text{D.O.C.}) \times (\text{W.O.C.})} = \frac{80}{.25 \times 2} = 160$$

Answer: = 160 IPM

12.6.1 Direction of Milling Feed

The application of the milling tool in terms of its machining direction is critical to the performance and tool life of the entire operation. The two options in milling direction are described as either conventional or climb milling. Conventional and climb milling also affects chip formation and tool life as explained below. Figure 12.18 shows drawings of both conventional and climb milling.

Conventional Milling: The term often associated with this milling technique is 'up-cut' milling. The cutter rotates against the direction of feed as the workpiece advances toward it from the side where the teeth are moving upward. The separating forces produced between cutter and workpiece oppose the motion of the work. The thickness of the chip at the beginning of the cut is at a minimum, gradually increasing in thickness to a maximum at the end of the cut.

Climb Milling: The term often associated with this milling technique is 'down-cut' milling. The cutter rotates in the direction of the feed and the workpiece, therefore advances towards the cutter from the side where the teeth are moving downward. As the cutter teeth begin to cut, forces of considerable intensity are produced which favor the motion of the workpiece and tend to pull the work under the cutter. The chip is at a maximum thickness at the beginning of the cut, reducing to a minimum at the exit. Generally climb milling is recommended wherever possible. With climb milling a better finish is produced and longer cutter life is obtained. As each tooth enters the work, it immediately takes a cut and is not dulled while building up pressure to dig into the work.

Advantages and Disadvantages: If the workpiece has a highly abrasive surface, conventional milling will usu-

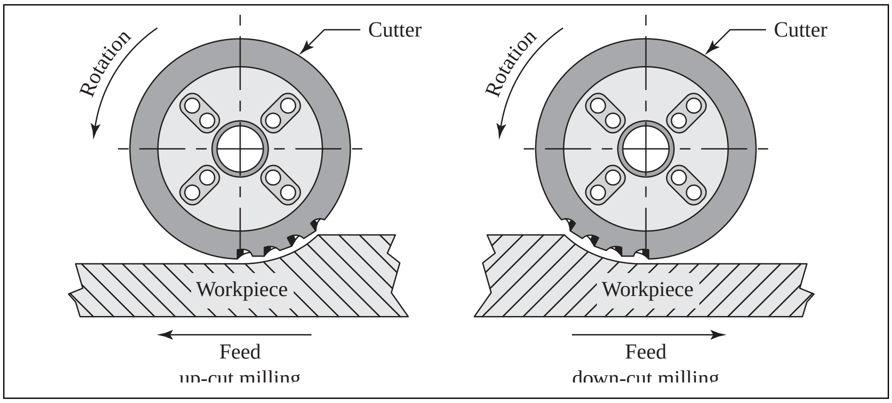


FIGURE 12.18: Conventional or up-milling as compared to climb or down-milling.

ally produce better cutter life since the cutting edge engages the work below the abrasive surface. Conventional milling also protects the edge by chipping off the surface ahead of the cutting edge.

Limitations on the use of climb milling are mainly affected by the condition of the machine and the rigidity with which the work is clamped and supported. Since there is a tendency for the cutter to climb up on the work, the milling machine arbor and arbor support must be rigid enough to overcome this tendency. The feed must be uniform and if the machine does not have a backlash eliminator drive, the table gibs should be tightened to prevent the workpiece from being pulled into the cutter. Most present-day machines are built rigidly enough. Older machines can usually be tightened to permit use of climb milling.

The downward pressure caused by climb milling has an inherent advantage in that it tends to hold the work and fixture against the table, and the table against the ways. In conventional milling, the reverse is true and the workpiece tends to be lifted from the table.