Objectives

Information in this chapter will enable you to:

- List several safety precautions to observe while machining with carbide inserts.
- List four milling cutter types used in machining.
- Identify different cutter mounting methods used for milling cutters.
- Differentiate between a negative and a positive insert entry angle.
- Differentiate between a 0° and a 45° lead angle.
- Recognize the different insert shapes.
- Explain the milling insert identification system.
- Explain how the size of an insert is specified.

Technical Terms

- Adapter mounting
- Collet chuck adapter mounting
- End mill adapter mounting
- Insert grades
- Insert size
- Lead angles
- Nose radius
- Weldon Shank

Milling

Milling is the process of removing material with a multitooth rotating cutter. Milling involves a nonrotating workpiece, interrupted tool engagement, varying chip thickness, and varying cutting forces. An operator should possess a working knowledge of cutter geometry, carbide grades, setup rigidity, and horsepower requirements. The four basic types of milling cutters are end mills, face mills, slotting cutters, and thread mills, Figure 6-1.

Cutter Mounting

Milling cutters are mounted into the spindle of a CNC machine with arbors or adapters. Three commonly used methods of cutter mounting are adapter mounting, end mill adapter mounting, and collet chuck adapter mounting.

- Adapter mounting. With adapter mounting, the cutter is mounted on a piloted adapter that uses drive keys. The cutter is held in place by a lock screw or socket head cap screw. See Figure 6-2. This mounting is used for cutters six inches or less in diameter.
- End mill adapter mounting. In end mill adapter mounting, the cutter is held with a lock screw against a Weldon Shank. This mounting is used for general-purpose work. See Figure 6-3.
- Collet chuck adapter mounting. With collet chuck adapter mounting, the collet reduces runout. This provides better finish and size control. See Figure 6-4.
In addition to these three methods of holding cutters, there are many other types of holders for cutters. More information about these variations can be obtained from tool manufacturers’ catalogs. If you are purchasing or using these types of tools, you should spend time familiarizing yourself with the many types of cutters and holders available.

Safety Precautions

Certain safety precautions should be taken when machining materials with carbide inserts. These precautions include:

- Protect against burns or physical injury that could result from high-temperature chips coming off the workpiece.
- Select an insert size and shape that is adequate for the job.
- Avoid continuous chips that could become entangled with the tool and work.

Safety Precautions

- Do not remove chips by hand because of their high temperature and sharp edges.
- Do not use an air hose to blow chips from the machine or your clothing.
- Keep tool overhang to a minimum to avoid tool chatter and possible insert breakage.
- Take extreme care to ensure both the tool and workpiece are tight and secure.
- Keep cutting fluids clean to avoid the possibility of foreign particles interfering with the workpiece finish.
- To avoid the possibility of igniting the coolant because of high temperatures generated during cutting, select the proper coolant and use sharp tooling.
- Do not overload the carbide inserts with excessive pressure; this could lead to insert breakage.
- Be careful when machining aluminum, magnesium, and titanium; they represent a potential fire hazard.
- Always use the largest size tooling available.

Cutter Selection

There are a number of factors to consider when selecting a cutter. These factors include cutter size, insert entry angle, milling cutter pitch, and lead angles.

Cutter Size

The size of the workpiece best determines the size of the cutter that should be used. A rule of thumb is that the cutter should be approximately one and one-half times the width of the workpiece. If the width is greater than any cutter size available, multiple passes should be made. With multiple passes, one-fourth the width of the cutter body should be outside the workpiece when making the first pass.

Insert Entry Angle

A negative entry angle is recommended to absorb the shock of the insert entering the cut. The strongest part of the insert is near the cutter body. A positive entry causes the tip of the insert to absorb the shock of entry. This could lead to the insert breaking.

Milling Cutter Pitch

Cutter pitch refers to the number of inserts found on a milling cutter. A coarse pitch is used for general-purpose milling where maximum depth is required and adequate horsepower is available. A medium pitch is used
when it is necessary to have more than one insert in contact with the work. A fine pitch is used on workpieces that present an interrupted cut condition. Increasing the number of inserts on a cutter allows the feed rate (ipm) to be increased.

**Lead Angles**

Lead angles on a milling cutter have an effect on cutting force direction, chip thickness, and tool life. Lead angles can be 0°, 15°, 20°, and 45°. A 0° lead is used to produce a 90° shoulder. See Figure 6-5. For general milling applications where rigid conditions exist, 15° and 20° leads are used. A 45° lead angle allows higher feed rates to be used. As the lead angle increases from 0° to 45°, the amount of entry shock minimizes. The lead angle will affect chip thickness. Figure 6-6 shows that as the lead angle changes, the chip thickness changes as well.

**Insert Selection**

There are a number of factors that help determine which insert to use. These factors include insert grade, insert shape, insert size, and nose radius.

**Insert Grade**

Carbide insert grades are based on their toughness and their ability to resist wear. A manufacturer’s technical guide should be used to select the proper grade needed to machine different materials. Different grades are also based on the type of cuts being taken. Systems for grading carbide inserts based on the insert’s application and its physical characteristics have been developed by the American National Standards Institute (ANSI) and the International Standards Organization (ISO).

Charts recommending the grades to be used for different machining conditions are available from manufacturers. There are also cross-reference charts available in catalogs to help make selections among different manufacturers’ grades. In a case where a certain grade is not available from one manufacturer, these charts can be used to select a similar grade from another manufacturer.

**Insert Shape**

Different insert shapes accomplish specific types of machining operations on various workpiece shapes. See Figure 6-7. The strength of the insert also depends on the shape of the insert. For example, the round insert has the greatest strength and the most cutting edges, while the 35° diamond insert has the lowest strength. The triangular insert is very versatile for milling operations.

---

**Figure 6-5.** An example of an insert with a 0° lead angle, which is used to create a 90° shoulder. (Kennametal)

**Figure 6-6.** These charts show that a larger lead angle results in a thinner chip when the feed remains the same.

**Figure 6-7.** These are the basic carbide insert shapes. Strength of the insert increases as corners increase in angular value. The round insert is strongest. (Kennametal)
Insert Size

Insert size is determined by the largest inscribed circle (IC) that will fit inside the insert or touch all edges of the insert, Figure 6-8. The most common insert is a ¾" square. As the size of the insert increases, the depth of cut can increase as well. A rule to follow is to set the depth of cut to no more than ½ the cutting edge length.

Nose Radius

The nose radius, Figure 6-9, will affect tool strength and surface finish. The larger the nose radius, the stronger the insert and the better the finish. However, if tooling and setup are not rigid, a large nose radius can cause chatter. The chart in Figure 6-10 indicates the smoothness of surface finish that should result from various combinations of nose radii and feed rates.

Insert Identification System

A standardized letter and number system developed by ANSI, Figure 6-11, is used to identify and classify indexable carbide inserts. The identification reference is a combination of nine letters and numbers. The reference indicates the insert shape, clearance (relief angle), tolerance, insert type, size, thickness, point (nose) radius or chamfer, cutting edge condition, and (optional) manufacturer’s identifier.

Figure 6-8. The diameter of an inscribed circle determines the size of the insert. In this installation, IC is the imaginary inscribed circle, B represents the distance from a tangent of the circle to the corner of the insert, and T is the thickness of the insert. (Kennametal)

Figure 6-9. A large nose radius value results in a better finish, but can also cause chatter if tooling and setup are not rigid. (Kennametal)

Figure 6-10. This metal finish chart predicts the smoothness that will result from different insert nose radius and feed rate combinations. For example, a 1/8" radius and .010" feed results in 25 rms finish, while a 1/32" radius and .008" feed results in 60 rms finish. (Kennametal)
Preparation for Machining

A machinist should be aware of the following items before starting a machining process.

- Check toolholders and replace any missing or damaged parts.
- Check the size of milling cutters, especially if reground.
- Check cutting edges of tooling for sharpness. Regrind if they are dull, nicked, or broken.
- Use the largest toolholder available for maximum rigidity.
- Use the shortest tool and toolholder for maximum rigidity.
- Use proper speeds and feeds for the type of tooling and the operation being performed.
- Use the correct cutting speed for the work material and tool material selected.
- Use a sufficient number of clamps and straps to fasten work to the table.
- Be sure the workpiece is sufficiently placed in a vise for maximum gripping power.
- Before making cuts, study the part drawing carefully to avoid mistakes.
- Set up clamps close to the workpiece and keep them parallel with the table.
- Always use parallels when holding the workpiece in a vise.
- Use the strongest inserts possible, depending on the type of cuts to be made.
- Use the maximum depth of cut and a coarse feed for roughing.
- Direct cutter force against the solid jaw of the vise.
- Remove all burrs and sharp edges from workpieces before fastening.
- Use proper coolant to prolong tool life.

Figure 6-11. This graphic identifies the meaning of each letter or number in the standard identification system used to classify carbide inserts. The system was developed by the American National Standards Institute (ANSI).
Summary

Milling is the process of removing material with a multitooth rotating cutter. There are four basic types of milling cutters. Cutters are mounted on adapters, end mill adapters, and collet chucks. Cutters are selected based on size, insert entry angle, insert pitch, and lead angles. Inserts are selected on the basis of grade, shape, size, and nose radius. A standard letter and number system is used to identify and classify cutting inserts.

Chapter Review

Answer the following questions. Write your answers on a separate sheet of paper.

1. How do you avoid the possibility of igniting coolant as a result of high temperatures generated from cutting?
2. List the four basic types of milling cutters.
3. What type of entry angle is recommended to absorb the shock of a carbide insert entering the machining cut?
4. What is pitch?
5. What milling cutter lead angle is best for allowing higher feed rates to be used?
7. When dealing with inserts, what do the letters IC mean?
8. What do the letters CNMG mean when used to identify an insert?
9. What do the numbers 432 mean in the insert designation CNMG-432?
10. List five precautions that should be taken while preparing to machine a workpiece.

Activities

1. Obtain a sampling of the various types of inserts available based on shape and size, and arrange a display.
2. From a part drawing, select inserts best suited to manufacture the part.
3. Use an insert troubleshooting chart that shows causes and remedies for insert problems, and list ten problems that can arise from machining.
4. Obtain catalogs from manufacturers of cutting tool holders and inserts and develop a library of machining data.

In order to identify each of the many varieties of carbide inserts, part numbers are printed directly on the inserts. (Seco Tools AB)