# Introduction to Milling Tools and their Application <u>Identification and application of cut</u>ting tools for milling



The variety of cutting tools available for modern CNC milling centers makes it imperative for machine operators to be familiar with different types of milling cutters and how they are applied to everyday milling processes.

This course curriculum contains 16-hours of material for instructors to get their students ready to identify different types of milling cutters and their uses.



# Table of Contents

Introduction
Audience2
Purpose2
Lesson Objectives
Anatomy of a Milling Tool
End Mills
Flutes5
Applications for End Mills6
End Mill Materials7
End Mill Coatings7
Face Mills and Indexable End Mills
Drills, Taps and Reamers9
Twist Drills9
Indexable Drills
Spot Drills and Center Drills 10
Reamers
Taps
Tool Holders
End Mill Holders
ER Collet Chuck
Milling Chuck
Drill Chuck
Hydraulic Holders and Shrink Fit Holders14
Feeds and Speeds15
Cutting Speed 15
Feed Rate16
Selection of Tools, Feeds, and Speeds16

# Introduction

Milling produces 3-dimensional shapes with a rotating multi-edge cutting tool. The cutting tool can be programmed to move against a fixed workpiece in almost any direction. Milling tools remove material by their movement in the machine and from their shape.

#### Audience

This class is intended for entry-level milling operators and students in a milling operator training program who have a basic understanding of milling machines and their operation. This lesson is also useful to anyone interested in the metalworking industry who wants to gain knowledge about material removal in CNC machining.

#### Purpose

Learn how to visually identify the types of cutting tools most commonly used in CNC milling operations and how those tools are used in everyday milling processes. Students are introduced to tools for milling and hole making operations, and common tool holder configurations. Students finish with an introduction to the concepts of spindle speeds and feed rates.

#### Lesson Objectives

At the end of this lesson, you will know how to:

- Identify common types of tools for milling
- Identify common types of tools for drilling and threading
- Identify basic tool holders for milling
- Identify the main groups of cutting tool materials and coatings
- Understand the applications for common milling tools
- Understand the difference between cut speed and feed rate
- Understand the difference between Revolutions Per Minute (RPM) and Surface Feet per Minute (SPM)
- Understand the difference between Inches per Minute (IPM) and Inches per Tooth (IPT)

# Anatomy of a Milling Tool

Milling is done using a cylindrical milling tool mounted in a milling tool holder that is then mounted in the tool spindle on the machine.



# End Mills

End mills are the most common milling cutters. End mills are available in a wide variety of lengths, diameters, and types.

A square end mill is used for most general milling applications. It produces a sharp edge at the bottom of pockets and slots.



End mills can be center cutting and non-center cutting. As their name implies, center cutting end mills have cutting edges on both the end face of the cutter and the sides. Center cutting end mills are essential for plunge milling.



Non-center cutting end mills have cutting edges only on the sides and are used only for side milling. These tools are identified by a small hole at the center.



Roughing end mills have serrations in the teeth to quickly remove large amounts of material without creating vibration. The serrations produce many small chips and a rough finish.



Ball end mills produce a radius at the bottom of pockets and slots. Ball end mills are used for contour milling, shallow slotting, contour milling and pocketing applications.



#### Flutes

Spiral-shaped cutting edges are cut into the side of the end mill to provide a path for chips to escape when an end mill is down in a slot or a pocket.

The most common options are 2, 3, or 4 flutes. The more flutes, the more cutting edges but the narrower the channel for chip evacuation.



An important consideration when choosing the number of flutes is called "chip load".

"Chip load" is a measurement of the thickness of material removed by each cutting edge during a cut.



The number of flutes you choose depends on the material you want to cut and the capabilities of your machine.

A material like aluminum produces large chips compared to other materials. For that reason, 4 flute end mills are rarely used with aluminum because the flutes can get jammed with chips and break the cutter.

For harder materials, you want to use more flutes. Having more flutes reduces chip load and improves surface finish.

While the number, direction and type of flutes that a cutting tool has can vary widely, the tools most commonly used have two flutes and are up-cut spirals to move the chips up out of the cut.

- **Two Flute**: Has the greatest amount of flute space, allowing for more chip carrying capacity in softer materials. Used primarily in slotting and pocketing of non-ferrous materials like aluminum where chip removal is a concern.
- **Three Flute**: Allows for better part finish in harder materials. The three flutes provide for greater strength and the ability to pocket and slot both ferrous and non-ferrous materials.
- Four Flute/Multiple Flute: Ideal for finish milling. The extra flutes allow for faster feed rates to produce a much finer finish than two or three flute tools. However, the reduced flute space may cause problems with chip removal.

The most common flute numbers for general milling operations are two (better space for chip ejection) and four (better surface finish).

Side Milling	Face Milling	Slot Milling	Plunge Milling	Ramping
Machining of an edge surface on the part.	Machining of a top face on the part.	Machining between two edge surfaces.	Axial feeding into a part along the Z axis. Requires a center cutting end mill.	Axial feeding into a part along the Z axis as well as X or Y. Requires a center cutting end mill.

#### Applications for End Mills

# **End Mill Materials**

End mills are made out of either cobalt steel alloys (known as high speed steel, or HSS), or from tungsten carbide in a cobalt lattice (shortened to "carbide").

- **High Speed Steel (HSS)**: Provides good wear resistance and costs less than cobalt or carbide end mills. HSS is used for general purpose milling of both ferrous and non-ferrous materials. While usually inexpensive, HSS does not offer the tool life or speed advantages of cobalt and carbide end mills.
- **Cobalt**: Cobalt is an M42 tool steel with an 8% cobalt content. Cobalt is more expensive but provides better wear resistance and toughness than HSS (M7). Because the tool can run 10% faster than HSS, metal removal rates and finish are better than HSS.
- Solid Carbide: Carbide is considerably harder, more rigid, and more wear resistant than HSS. However, carbide is brittle and tends to chip instead of wear. Carbide is used primarily in finishing applications. Carbide tools are best suited for shops operating newer milling machines or machines with minimal spindle wear. Rigidity is critical when using carbide tools. Carbide end mills may require a premium price over the cobalt end mills, but they can also be run at speeds 2 1/2 times faster than HSS end mills.

The choice of tool material depends on the material to be cut as well as on the maximum spindle speed of the machine. Smaller milling machines may not be capable of reaching the spindle speeds recommended for carbide end mills.

# **End Mill Coatings**

The use of coatings will increase the surface hardness of the tool. This will allow for greater tool life and faster cutting speed.

Standard coatings include Titanium Nitride (TiN), Titanium Carbonitride (TiCN), and Aluminum Titanium Nitride (AlTiN).

Long-life TiN (titanium nitride) coating is good for use on alloy steel, aluminum, and plastic. Color is gold.



Extra-life TiCN (titanium carbonitride) coating has better wear resistance than TiN coating, making it a good choice for tough-to-machine materials such as ductile cast iron, stainless steel, aluminum, and plastic. Color is blue-gray.



Super-life AlTiN (aluminum titanium nitride) coating is the best for very high feeds/speeds and high-temperature applications. Use to mill cast iron, stainless steel, nickel-based alloys, and titanium. Not for use on aluminum. Color is purple-gray.



The choice to use coated end mills is a cost versus benefit choice. If your machining benefits enough from the extra performance of a premium coating, purchase end mills with a coating.

### Face Mills and Indexable End Mills

Face mills are tools with a large diameter that are used to cut a wide shallow path for facing operations. Facing is used for machining a large flat area, typically the top of the part in preparation for other milling operations.

A face mill is a solid body that holds multiple carbide inserts that can be replaced as they wear out. The more inserts, the faster the metal removal rate.

Cutters with a 45° lead angle are used most frequently.



Square shoulder (90°) and round insert cutters are also used for certain conditions.



Indexable end mills also use replaceable carbide inserts. Indexable end mills are convenient for efficient roughing and to save costs on the larger sizes of solid end mills. Indexable end mills are typically used to rough the part, which is then finished with a solid end mill.



### Drills, Taps and Reamers

High Speed Steel (HSS) tools such as drills, taps and reamers are commonly used on CNC machining centers for hole making operations.

#### **Twist Drills**

Holes are by far the most common feature in CNC machining. The material removal rate of twist drills is outstanding, and usually better than equivalent sized end mills. Twist drills come in many more diameters and sizes than end mills. It often makes sense to drill the ends of a slot and then machine out the web between the holes with an end mill.



Twist drills have a conical cutting point at the tip of a cylindrical shaft that has one or more helical flutes. The flutes are designed to evacuate chips out of the hole being drilled. Twist drills

are available in a variety of sizes, lengths, materials and coatings, but the most common are High Speed Steel (HSS) and solid carbide.

Tool coatings have an influence on the cutting process by increasing cutting speed and tool life. Coatings such as titanium nitride (TiN) increase the initial cost but reduce wear and increase tool life. Applied as a thin coating, TiN is used to harden and protect cutting surfaces.

Twist drills coated with titanium nitride (TiN) are easily identified by a gold like color. This coating increases the hardness of the bit and adds a self-lubricating property.



#### Indexable Drills

Considering how often holes are made and the advantages of indexable tooling for end and face milling, indexable drills are also available. These are generally best used for larger holes.



#### Spot Drills and Center Drills

These two are special purpose drills with very short flutes.

Spot drills are designed to be extremely rigid so that they can precisely spot a hole for a twist drill. The goal is use the spot drill to make a little dimple in the workpiece that keeps the twist drill from walking so that the hole winds up in the right place.



Center drills are intended to be used to create a 60° center in the end of lathe stock. They have a 2-part tip that has a small pilot as well as the larger countersinking area of the bit. In theory,

there is no reason to use a Center Drill on a mill. Their secondary "pilot" tip makes them more delicate than spot drills. However, many machinists will grab one anyway if it is handy and use it as they would a spot drill.



If you use a carbide drill, spotting is typically not needed. The carbide itself is so rigid compared to HSS that the drill will go where it is pointed.

#### Reamers

Reamers are used to enlarge an existing hole to a precise tolerance and to add a high quality surface finish. Reamers require a hole be drilled first that is fairly close to the final size so that the reamer actually removes relatively little material. Reamers ensure a hole has an accurate diameter, roundness, and good surface finish.



#### Taps

Taps are used to cut internal threads of a specific size and pitch. Like reamers, a tap requires a hole be drilled first to the size of the minor diameter.



Not all threads are made by cutting the material. There are two major categories of taps: Roll Form and Cut Taps. Unlike thread cutting, a Roll Form tap does not produce any chips. Instead of cutting, the tap is forced into the hole and the material is deformed into the required thread form. This process creates stronger threads and reduces manufacturing time.

Forming taps must be applied in materials that cold form well. This includes steels, stainless steels, light metals, and light metal alloys. Generally, materials that produce a continuous chip when drilling are good candidates for thread forming.

# **Tool Holders**

Tool holders are used to adapt tools of different shapes and sizes accurately and securely to a milling machine spindle. Shanks are shaped to fit various standard machine tapers such as R8, NT 40, CAT 40, and BT 40.



Pull studs (retention knobs) are threaded into the tapered end of the tool holder so the holder can seat securely in the CNC machine.



The successful application of milling depends on how well the tool is supported by the tool holder. To achieve best results an end mill must be mounted concentric in a tool holder.

#### **End Mill Holders**

A solid end mill holder consists of a precision bored hole and a set screw. An end mill that has a shank with a Weldon flat is slid into the holder and then secured by the set screw from the side. The set screw clamps down on the Weldon flat, providing a strong grip with a slight loss in concentricity.

Solid end mill holders are often preferred for roughing and heavy duty milling because of their secure holding ability.



#### **ER Collet Chuck**

An ER collet chuck consists of a collet nut, a collet, and a tapered socket. Collet chucks have high accuracy and a wide range of gripping sizes, allowing the chuck to hold many different sizes and types of tools.

Collet chucks are often preferred for finishing because of their accuracy and versatility. Collet chucks have less holding security than solid end mill holders.



#### Milling Chuck

Milling chucks have a greater gripping power than traditional collet chucks and are more rigid and accurate. Milling chucks are preferred for larger milling tools and high-speed applications



#### **Drill Chuck**

Drill chucks are convenient and versatile for general hole drilling. ER collet chucks can also be used to hold drills that do not need to be switched out often.



#### Hydraulic Holders and Shrink Fit Holders These two are specialty holders for high-speed applications.

Hydraulic holders use fluid to center and compress the tool with uniform pressure, allowing for outstanding concentricity and rigidity of the tool during machining.

Shrink fit holders use thermal expansion to center the tool. The diameter of the bored hole is intentionally undersized so that it can expand sufficiently to accommodate the tool after it is heated in an induction heater. Cooling of the holder then creates the clamp when the metal contracts.

# Feeds and Speeds

Feeds and speeds refer to two separate velocities for machine tools: feed rate and cutting speed. They are used together because of their combined effect on the cutting process.

- Cut speed is the speed at the outside edge of the milling cutter as it is rotating
- Feed rate is the velocity at which the cutter is advanced along the workpiece; its vector is perpendicular to the vector of cutting speed

#### **Cutting Speed**

Cutting speed is a velocity unit expressed in terms of revolutions per minute or surface feet per minute. This is the speed that the cutting edge moves past the material.

Revolutions Per Minute (RPM) relates directly to the speed, or velocity, of the tool spindle. It annotates the number of turns completed in one minute around a fixed axis. RPM maintains the same revolutions per minute throughout the entire operation.

Surface Feet Per Minute (SFM) is a combination of the tool diameter and RPM. The faster the spindle turns, and/or the larger the tool diameter, the higher the SFM. If two cutters of different sizes are rotating at the same revolutions per minute, the cutter with the larger diameter results in a greater cutting speed because it has a larger circumference and has more surface area.

Materials will run better at specific SFMs. SFM is a constant, with RPM as a variable based upon tool diameter.

When the SFM constant is known for a specific material, the formulas below can be used to determine spindle speed for milling various materials.

RPM = SFM x (12/Pi) / Tool Diameter

So, for an SFM of 400 and a tool diameter of 1/2", RPM can be found:

RPM = 400 x 3.82 / 0.5 = 3056 RPM

The calculation of SFM is the opposite:

SFM = RPM x Tool Diameter x Pi/12

For example, if you have a 1/2" tool and a spindle speed of 3056 RPM, then:

SFM = 3056 x 0.5 x .262 = 400

#### Feed Rate

Feed rate is the velocity at which the cutter is advanced along the workpiece. Feed rate is expressed as units of distance per minute or distance per tooth (chip load).

Feed rate can be defined as Inches Per Minute (IPM) or Inches Per Tooth (IPT). IPT, or chip load, is more commonly used.

Values for IPM and IPT are easily converted with the following formulas:

IPM = RPM x IPT x Number of Flutes

IPT = IPM / RPM / Number of Flutes

So, for a spindle speed of 3056 RPM and a feed rate of .005 IPT with a 2 flute cutter, the IPM can be calculated as follows:

IPM = 3056 x .005 x 2 = 30 IPM

#### Selection of Tools, Feeds, and Speeds

Cutting tool selection has a direct impact on the proper programming of feeds and speeds at the machine.

However, many other variables that affect feeds and speeds are:

- Workpiece material class and condition
- Workpiece diameter
- Cutter material
- Cutter geometry
- Type of cut
- Depth of cut
- Condition of the machine

Cutting tool manufacturers publish the general feeds and speeds and recommended usage for the application. Cutting tool manufacturers are often a good place to start for recommendations on tool selection and feeds/speeds since they rely on customer loyalty. The customer (or potential) should select a cutter and cutter material based on the vendor's recommendation.