Enco Introduction to End Mills

End Mills are used for milling, profiling, contouring, slotting, counterboring, drilling and reaming.

CUTTING TOOL MATERIALS

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.

Cobalt (M-42: 8% Cobalt): Provides better wear resistance, higher hot hardness and toughness than high speed steel. There is very little chipping or microchipping under severe cutting conditions allowing the tool to run 10% faster than HSS, resulting in excellent metal removal rates as well as good finishes. This material, combined with the proper geometry has proven the most cost-effective for machining cast iron, heat treated steels and titanium alloys.

Powdered Metal (PM) Cobalt: A cost effective alternative to solid carbide, powdered metal is tougher and less prone to breakage. Tools perform well in materials <30RC. PM is used in high-shock and high-stock applications such as roughing.

Solid Carbide: Primarily used in finishing applications, carbide provides several advantages over HSS or cobalt materials. Carbide can run much faster because of the material's extreme hardness. This allows the cutter to withstand high cutting temperatures and provides excellent wear resistance. Carbide also provides better rigidity than HSS, which enables the end mill to provide a higher degree of dimensional accuracy and superior surface finishes. Carbide provides the user with the ability to run faster with less tool wear. However, the user should be aware that carbide's hardness comes at the expense of its toughness. Carbide is brittle and tends to chip when conditions are not ideal, rather than wear. Heavy feed rates are more suitable for HSS and cobalt tools. Carbide end mills, which can be run 2-3X faster than HSS, are best for maximizing speed and tool life.

Carbide-Tipped: Offering some of the advantages of solid carbide tooling, carbide is brazed to the cutting edges of steel tool bodies. This is particularly a cost-effective option for larger diameter tools.

END MILL TYPE

Finishing End Mills

Square End: Used for general milling applications

Ball End: Used for shallow slotting, contour milling and pocketing applications. Used to produce a radius in the bottom of slots or pockets for added strength. Also used to create compound curves for molds and dies where sharp corners must be avoided.

Roughing End Mills

In order to improve productivity over a conventional end mill, the roughing end mill (also known as a hog mill) was designed not to last longer, but to remove more material by taking heavier cuts in a fixed amount of time, without creating vibration. Ideal for today's rigid and powerful CNC machinery.

Coarse Tooth: This is the most commonly found profile. It is excellent in deep slotting conditions and heavy side cuts, and its design draws the least amount of power consumption. It is recommended for heavy machining conditions where high metal removal rates are required.

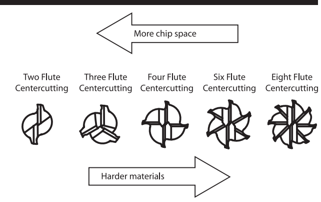
Fine Tooth: Designed to increase tool life over a coarser profile, this style is recommended for profiling operations. The fine profile has a shape, which provides the tool with a stronger edge that will last longer and permits a better surface finish on the workpiece. Works well in harder materials such as Inconel and Hastalloy.

FLUTE TYPE

Two Flute: The two-flute end mill has the greatest amount of flute space allowing for more chip carrying capacity. Used primarily in slotting and pocketing of non-ferrous materials where chip removal is a concern.

Three Flute: While this tool has the same flute space as two flutes, it has a larger cross-sectional area providing for greater strength and the ability to pocket and slot both ferrous and non-ferrous materials.

Four/Multiple Flute: Ideal for peripheral and finish milling. The additional flutes allow faster feed rates, but due to the reduced flute space, chip removal may be a problem. Produces a much finer finish than two and three flute tools.



END CUT TYPES

Centercutting: The tools have one or more cutting edges at the tip to allow the user to plunge, drill or ramp into a cut. This type of tool offers the user the greatest variety of applications.

Non-Centercutting: Peripheral teeth allow the user to side (radial) cut or contour an exterior surface. Used in applications where plunge cutting is not necessary.

SHANK TYPES

Plain Shank: The most common, used in conjunction with a collet type holder

Weldon Shank: The shank of the tool has 1 or 2 flats with set screws in holder that tighten on flanks of flats to hold the end mill, preventing it from rotating.

SURFACE TREATMENTS

TiN (Titanium Nitride): Provides extreme hardness and heat resistance allowing tools coated with TiN to run at higher speeds and feeds. Excellent for general-purpose use, TiN provides higher lubricity for freer chip flow, reduced buildup edge formation and catering. Use caution on non-ferrous materials because of a tendency to gall. Requires an increase of 25% to 30% in machining speeds vs. HSS.

TiCN (Titanium Carbonitride): Harder and more wear resistant than TiN under conditions of moderate cutting temperatures. TiCN can provide the users the ability to run the job at higher spindle speeds (50% over TiN), especially in stainless steels. Use caution on non-ferrous materials because of a tendency to gall. Requires an increase of 35% to 50% in machining speeds vs. HSS.

END MILL SELECTION SUMMARY

Select the shortest possible end mill for the greatest rigidity
Select two or three flutes for slotting or heavy stock removal
Select multiple flutes for finishing and greater rigidity
Use the largest diameter possible for added strength and rigidity
Use cobalt, PM/Plus, and carbide for tougher and harder materials, and for high production applications
Apply coatings for higher feeds, speeds, and tool life

MILLING FORMULA

RPM = SFM x 3.82 / Tool Diameter
IPM = RPM x # of Flutes x Chip Load
Chip Load = IPM / RPM x # of Flutes
SFM = .262 x Tool Diameter x RPM

Revolutions per Minute (RPM): How many revolutions the cutter has in one minute

Inches per Minute (IPM): Number of inches the cutter passes through the workpiece in one minute

Chip Load: The amount that each flute cuts during a single revolution of a cutting tool

Surface Feet per Minute (SFM): This is the cutting speed of the end mill in the United States. It is the number of feet per minute that a given point on the circumference of a cutter travels per minute