THE FUNDAMENTALS OF END MILLS:

An end mill will accurately remove workpiece material. Depending on the type of end mill, they are capable of slotting, grooving, plunging, ramping, facing, side-cutting, chamfering, tapering, making a convex or concave radius, contouring, profiling, and even threading.

FINISHING END MILLS

Ball End: Used for pocketing, contouring, profiling, radius grooving, and internal (concave) radii.

Chamfer Mills: Primarily used for chamfering, spotting and drilling.

Corner Radius: Used to increase End Mill performance, life, and rigidity, as well as creating a radius on the inside corner of a workpiece.

Corner Rounding: Used to machine rounded corners onto the edge of a workpiece.

Drill Mills: Primarily used for chamfering, spotting, drilling, countersinking and side milling.

Spherical Ball End: Used similar to a ball end mill, but with 220° of arc, rather than the 180° of a ball end mill.

Tapered: Used for machining a taper in a hole, pocket, or edge of a workpiece

Thread Mills: Used in a CNC to machine internal or external threads.

ROUGHING END MILLS

Roughing End Mills, also known as ripping cutters or hoggers, are designed to remove large amounts of metal quickly and more efficiently than standard end mills. A greater number of flutes reduces chip load and can improve surface finish, if feed and speed rates remain the same.

Coarse Tooth: remove large chips for heavy cuts, deep slotting and rapid stock removal on low to medium carbon steel and alloy steel prior to a finishing application.

Fine Tooth: remove less material but the pressure is distributed over many more teeth, for longer tool life and a smoother finish on high temperature alloys and stainless steel.

NUMBER OF FLUTES

Two Flute: Two flute end mills contain the greatest capacity of flute space for chip carrying capacity. The primary use of two flute end mills are the pocketing and slotting of non-ferrous materials where chip removal is needed.

Three Flute: Three flute end mills have less chip removal capacity than two flutes. However, the cross sectional area is larger which provides a greater amount of strength and the ability to slot and pocket both ferrous and non-ferrous materials.

Four/Multiple Flute: Having multiple flutes allows for faster feeds at the expense of chip removal due to the reduced area of flute space. However, this tool produces a finer finish than end mills with two or three flutes.

END CUT TYPES

Center Cutting:
This tool has one or more cutting edges at the tip, allowing the user to ramp or plunge into the workpiece. Center cutting also provides the greatest variety of applications.

Non-Center Cutting:
This tool is best used in situations where plunge cutting isn't needed. The peripheral teeth allows the user to contour or side cut an exterior surface.

Phone: **1.800.221.0270** | Fax: **1.800.722.0703** | Web: **www.travers.com** | Email: **sales@travers.com** TRAVERS

END MILL SHANK STYLES

Plain Shank: This type of shank must be held into the machine with a collet.

Weldon Shank: Contains 1 or 2 flats for set screws to tighten on the shank's flats in order to hold the end mill in place so it won't rotate or pull out of the end mill holder, and it can also be held in a collet.

FINISH/COATING

Bright Finish: This finish offers a more economical price point compared to coated tools.

AlCrN (Aluminum Chromium Nitride): Excellent wear resistance under conventional and extreme conditions when milling Die/Mold steels with a hardness 52 Rc and below. Excellent choice fortool steel, alloy steel, and stainless steel applications.

AlTiN (Aluminum Titanium Nitride): Increased thermal stability when milling high temp alloys and Die/Mold steels with a hardness 52 Rc and above. Excellent for HSM applications, Titanium, and Stainless Steels. HSS/HSCO end mills can't be coated with AlTiN.

TiN (Titanium Nitride): A general purpose coating for HSS, HSCO, and Solid Carbide end mills that provides effective protection against wear, abrasion, and edge buildup. Primary applications are milling steels in a non-hardened condition.

TiCN (Titanium Carbonitride): Incorporation of Carbon into the TiN matrix to increase hardness and abrasion resistance. TiCN is an alternative to TiN for HSS and HSCO applications where additional wear resistance is required. Primary solid carbide applications are milling aluminum alloys & cast iron.

TiAlN (Titanium Aluminum Nitride): TiAlN offers a higher level of abrasion resistance and thermal stability above that of TiN and TiCN. Ideal for high heat applications found in milling steels, stainless steels and high temp alloys with a hardness 52 Rc and below.

ZrN (Zirconium Nitride): ZrN provides a higher oxidation temperature than TiN, resisting sticking of heated aluminum, increasing lubricity and preventing edge build-up. ZrN also provides an improved finish when used in copper, aluminum, brass and other non-ferrous materials.

TOOL MATERIAL

Carbide Tipped: A cost effective option for tools that are larger in diameter. Offers similar advantages of solid carbide tooling (see below). The carbide is brazed to the cutting edges of steel tool bodies.

Cobalt: Usage results in little chipping when under severe cutting conditions. The tool is able to run 10% faster than high speed steel which allows for great metal removal rates and good finishes. This end mill also provides higher toughness, better wear resistance and higher hot hardness than high speed steel. It is also cost-effective for matching titanium alloys, machining cast iron and heat treated steels.

High Speed Steel: Commonly used for general purpose milling of both ferrous and non-ferrous materials. High speed steel costs less than carbide and cobalt while providing good wear resistance.

Powdered Metal (PM) Cobalt: Powdered Metal is a more economical alternative to solid carbide and is generally used in high-shock applications such as roughing. This tool is tougher and less prone to breakage than solid carbide.

Solid Carbide: Provides numerous advantages over cobalt materials and high speed steel. Solid carbide is more rigid than high speed steel and cobalt, enabling this end mill to display high degrees of dimensional accuracy, excellent wear resistance and great surface finishes. The strongest attribute of carbide is its hardness, which comes at the expense of its toughness. It should be noted that under non-ideal conditions, carbide tends to chip. Carbide is the best choice for maximizing speed and tool life since it can be run 2-3X faster than HSS, Cobalt, Carbide Tipped, or PM.

SELECTION TIPS

- Pick the shortest possible end mill for the greatest rigidity.
- Choose two or three flutes for slotting or heavy stock removal in softer materials.
- Pick multiple flutes for greater rigidity and finishing.
- For high production applications, use cobalt, PM, or carbide for tougher and harder materials.
- For higher feeds, speeds and tool life, apply coatings.

CHOOSING THE RIGHT HELIX ANGLE FOR YOUR MACHINING APPLICATION

The helix angle controls the angle of the cutting flute entering the workpiece as the tool rotates. The larger the helix angle, the more gradual the entry of the cutting flute. This creates a smoother transition as the cutting flute reaches the desired axial depth of cut.

HIGH VS. LOW HELIX ANGLES

THE PROS & CONS OF HIGH HELIX ANGLES

There are benefits to different degrees of helix angles as the changing of the angle will produce different results. A higher helix angles can result in reduced tool pressure, better finishes, and more efficiently dissipated heat. A 45° helix can increase downward pressure of a tool up to 50% vs. a 30° helix angle, which ultimately reduces load on the side of the cutter. The gradual entry of the cutting flute increases effectiveness when using advanced cutting tool programs such as trochoidal (volume) milling and high efficiency milling tool paths as the higher helix angles can be fed at a higher rate. A higher helix can also help with chip control by ejecting the chips at an angle. Proper chip evacuation can help prevent chip packing or re-cutting of chips in materials where we deliberately try to dissipate the heat on the chip itself. Higher helix angles are normally recommended on gummy materials and they perform well on aluminums, excel in stainless and heat resistant alloys. However, the larger the helix, the less reinforced strength each flute has. The high helix has been known to help pull the end mill out in collet fixturing (however most end mills pulling out of collet fixturing is due to user error or cutting parameters that are too aggressive).

THE PROS & CONS OF LOW HELIX ANGLES

Lower helix angles provide better edge strength in harder steels and cast irons. Tools with less than a 40° helix are typically more ideal for roughing applications due to their enhanced strength. Finishing quality, is however, compromised with these tool types. Cutting tools with a helix of 40° or lower are better suited only for finishing operations where the tool removes a minimal amount of material when running at higher velocities. They are not as strong as slow helix tools. Therefore, low helix tools are better suited for applications requiring heavier cutting, when finish is not a priority.

VARIABLE HELIX

Variable helix flutes also exist, where the flute will have multiple angles along the flute length, this is very helpful when contouring, trochoidal (volume) milling and high efficiency milling. The use of variable helix end mills can greatly reduce the chatter (noise & vibration), that is common in milling applications.

CARBIDE END MILL HELIX OVERVIEW

Helix Angle	**Recommended Workpiece Material**	**Example**
0°	Short chipping materials, abrasive plastics, epoxy-based synthetic materials, fiberglass, copper, brass, bronze and router set-ups.	
30°	Steels, cast irons, free machining stainless steels and roughing end mills.	
35°-40°*	Free machining low carbon steels, stainless steels & heat resistant super alloys.	
37°-45°	Non-ferrous soft materials such as aluminums, copper, plastics, etc.	
60°	Specialty applications for higher feed rates and better finishes.	
Variable	First choice for production or high-performance machining and for difficult to machine materials, such as stainless steels & heat resistant super alloys.	

Different manufacturers will utilize different angles (i.e. 37° vs. 38° for HRSA materials).

* Utilize a higher helix angle for free machining stainless steels and materials with lower machinability rate (more difficult to machine) such as Inconel, titanium, A286, etc.