makezine.com

The Skinny on End Mills | Make:

by Tyler Worman • Sept. 10, 2014 • 5 min read • original

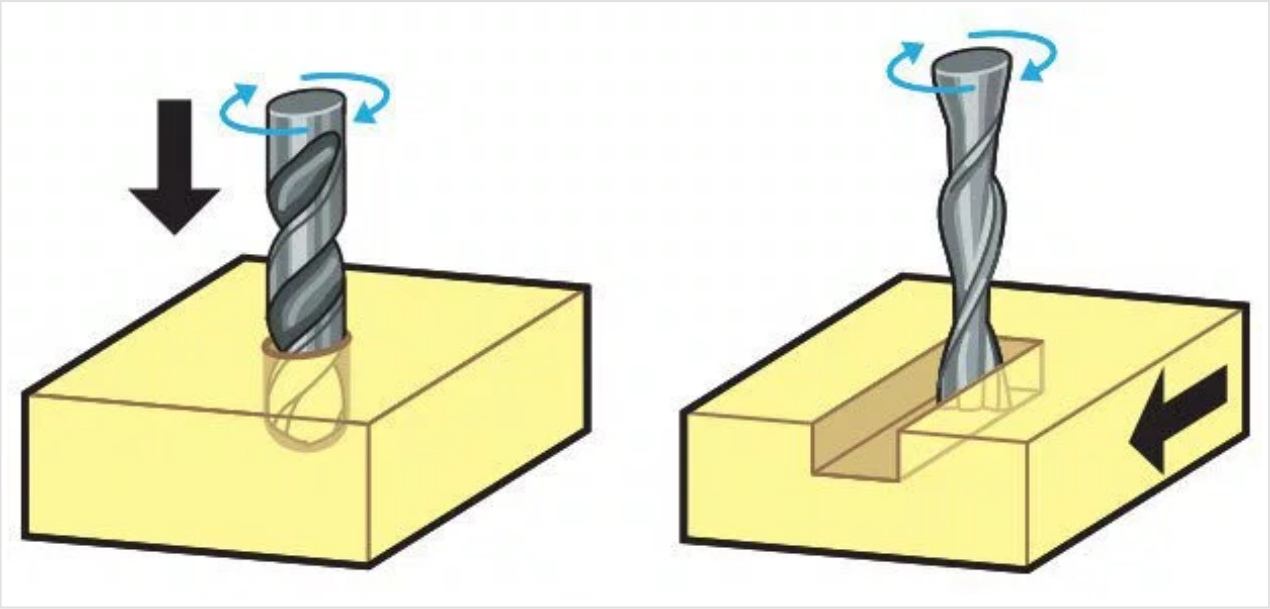
Interested in CNC routing but clueless about tooling? Can't tell an end mill from a drill bit? Here's an overview of end mill anatomy, some basic cutter types,

and tips on how to choose the correct tooling for basic wood or plastic jobs.

Drill Bits vs End Mills

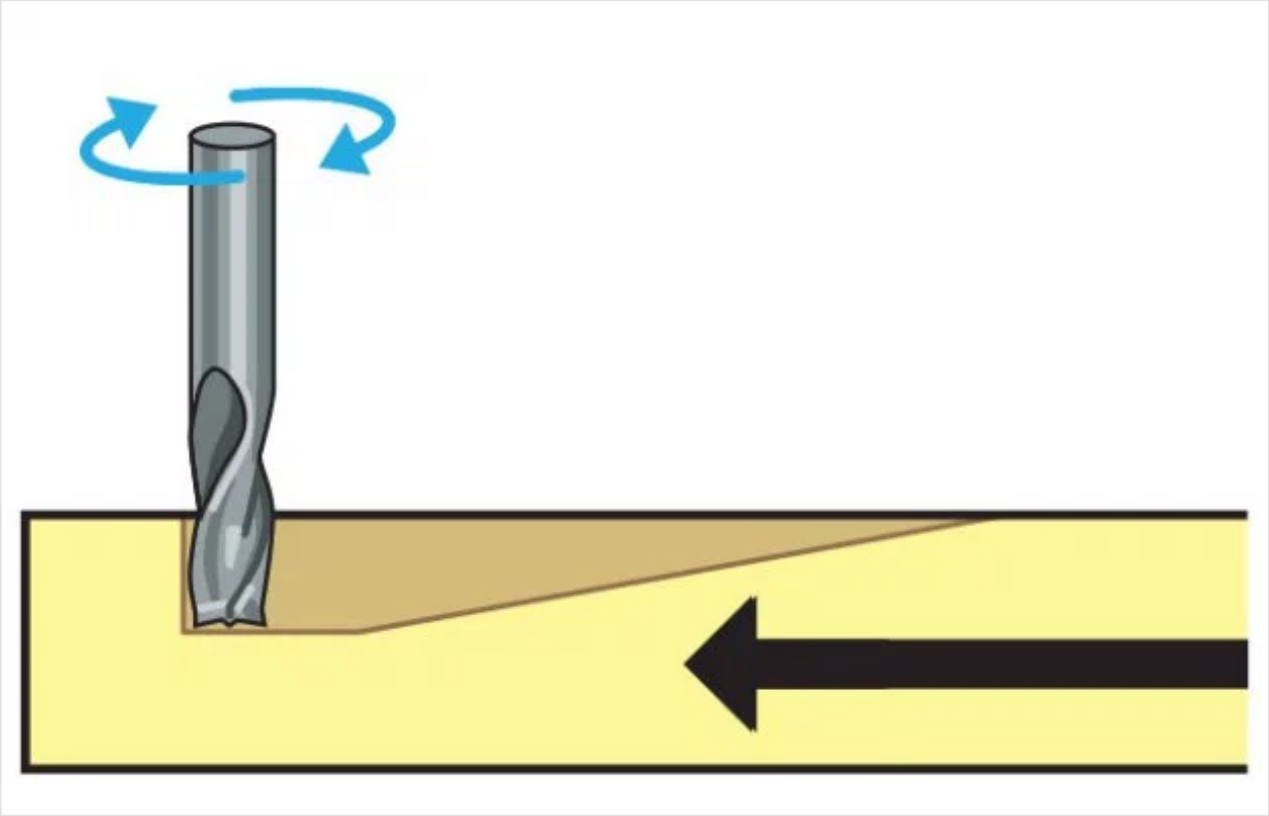
Top: Drill Bit, Bottom: End mill

CNC machining is a subtractive process that uses rotational cutting tools called “end mills” to remove material. An end mill, while similar in appearance to a drill bit, is far more versatile. However, in practice the terms “bit” and “end mill” are often used interchangeably.



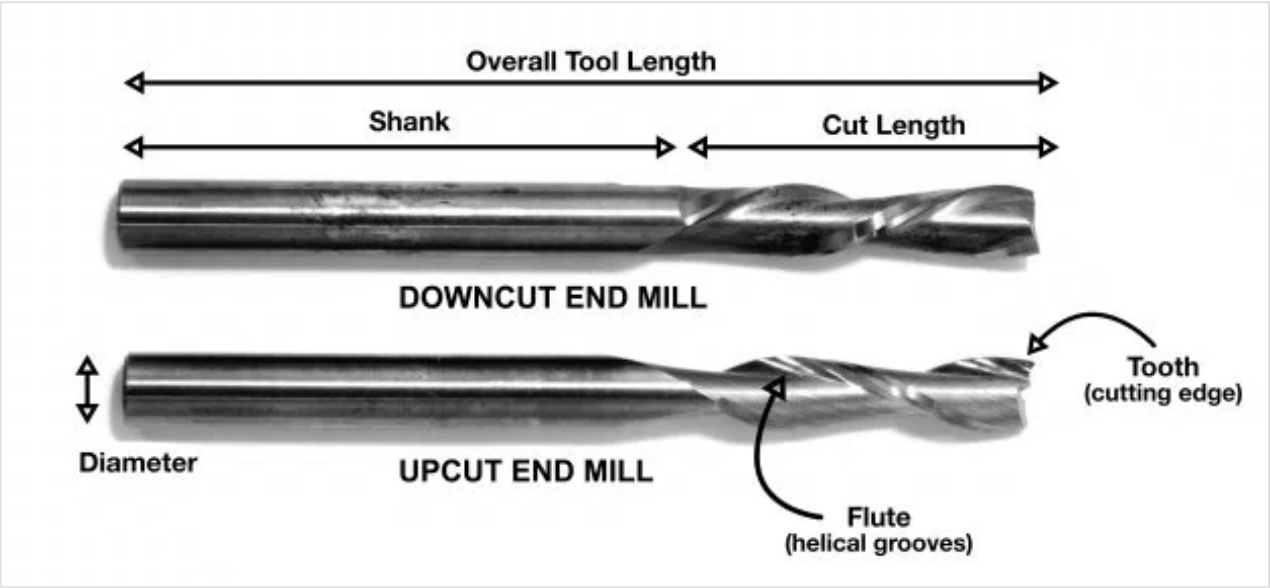
Drill plunging axially on left, endmill cutting laterally on right

Here's the key difference. Drill bits are designed to plunge directly into material, cutting axially and creating cylindrical holes. End mills are typically used for horizontal carving and cut laterally. Additionally, most mills are "center-cutting," meaning they are able to cut both axially and laterally. This is due to cutting flutes that extend to — and protrude from — the end face and enable plunge cutting. To minimize tool breakage and stress on the material being cut, most CNC software will "ramp" the end mill slowly into lateral cuts.



Endmill ramping into a cut.

The project type, material being cut, and desired surface finish determines the tool geometry. Key tooling features include the diameter, shank, flutes, teeth, tip shape, center cutting capability, helix angle, helix direction, length of cut, and overall tool length.

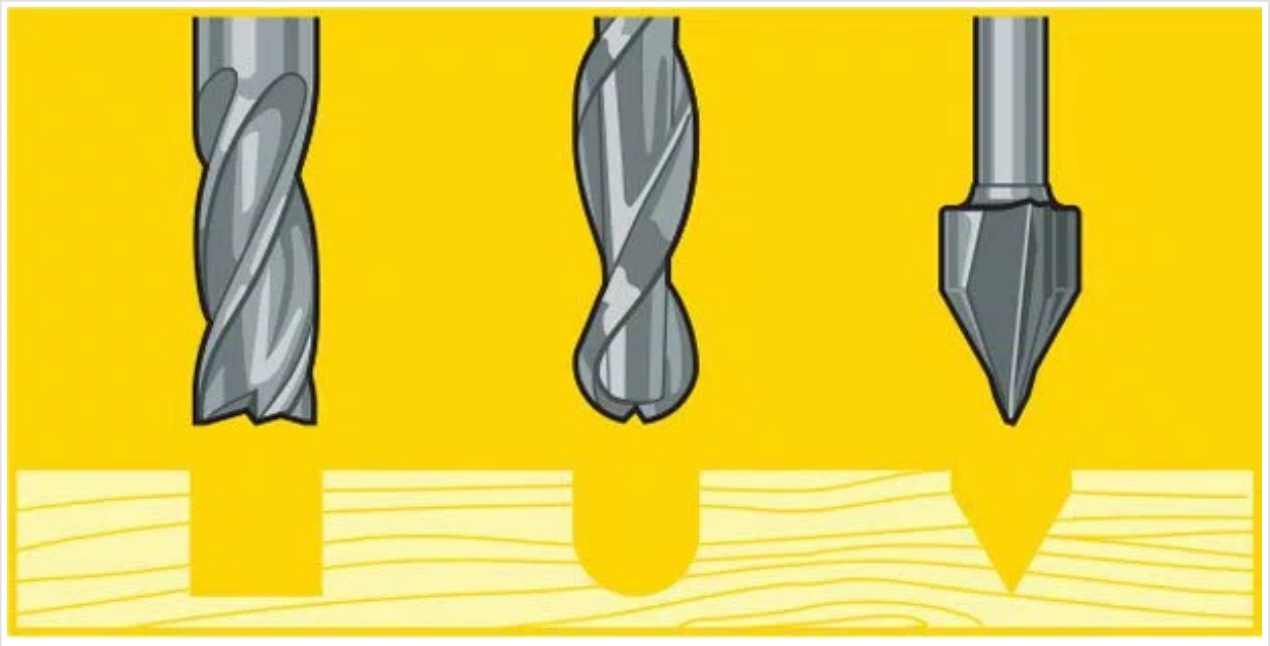


Basic End Mill Anatomy

Tip Shapes and Applications

Each end mill tip shape is designed for a particular purpose. Some common cutter shapes are ballnose, fish tail, surface planing, v-carving, and straight.

Ballnose mills produce a rounded pass and are ideal for 3D contour work, while fish tail cutters will produce a flat surface. V-bits produce a “V” shaped pass and are used for engraving, particularly for making signs.

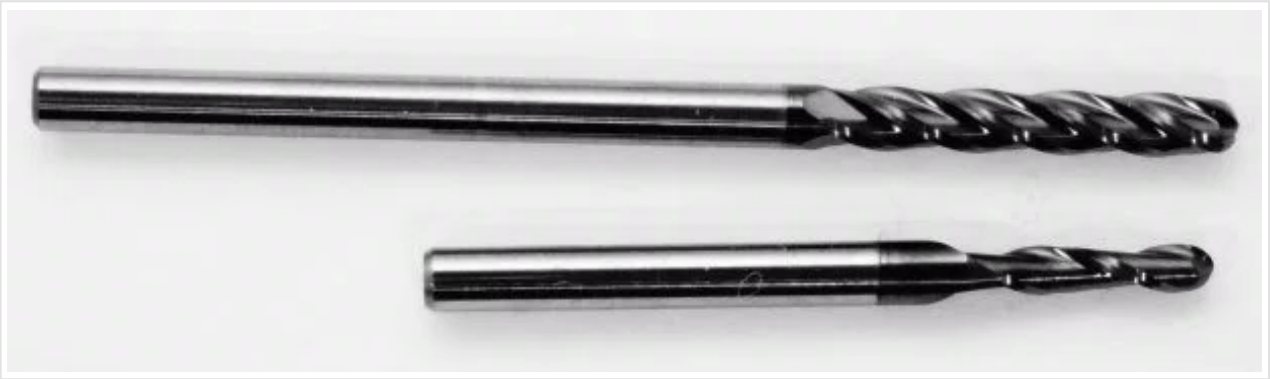


The difference in clearing path shapes between fish tail, ballnose and a v-carve mills.

The diagram above shows the difference in clearing path shape between a fish tail, ball nose and V tools. Ball nose mills are often selected when doing 3D contouring because their rounded edge reduces jagged steps when cutting several stepped layers. Ball nose mills can also be used to cut wide paths with rounded edges by reducing the step over amount (overlapping distance between) between passes. By overlapping steps, the central scallop shown in the diagram is eliminated.

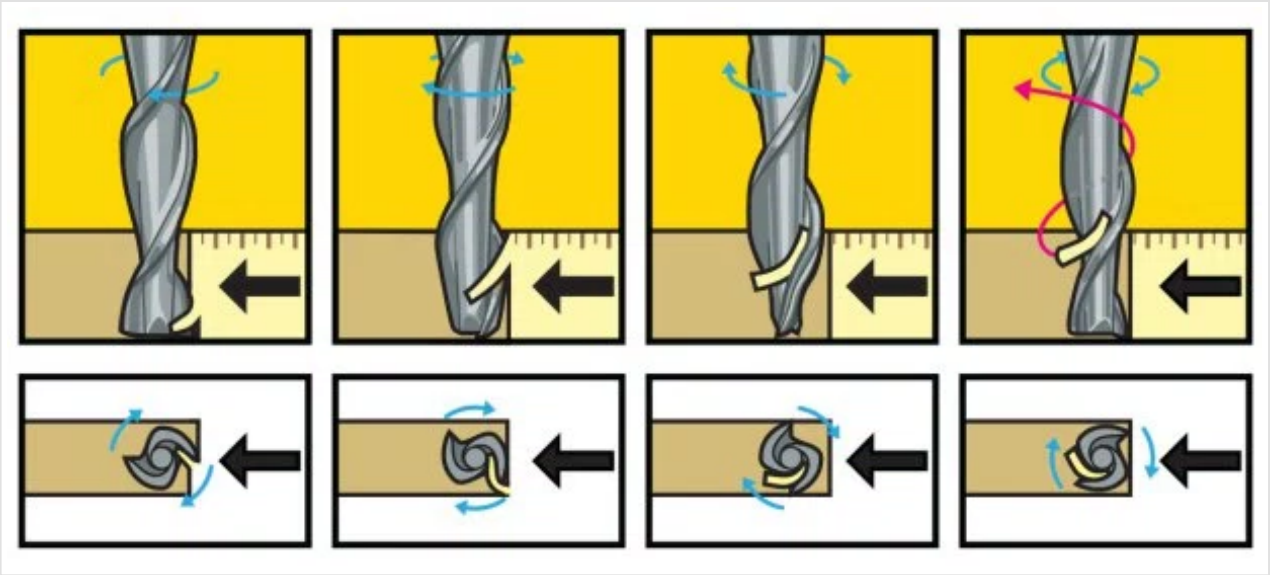
Flutes and Chipload

Flutes are the helical grooves that wrap around the sides of the end mill. Each flute has a single tooth with a sharp cutting edge (although there can be more than one) that runs along the edge of the flute.



TOP: Four flute endmill BOTTOM: two flute endmill

As the tooth cuts into the wood, each flute whisks away a small section or “chip”. The fewer the flutes, the more material that is ejected with each tool rotation. The overall cutting depth should never exceed the length of the flutes on an end mill. If cutting deeper than the length of the flutes, the tops of the flutes will be blocked and chips won’t clear, building up heat and reducing tool life.



Chipload is the thickness of a machined chip as cut by a specific tool type. More flutes create a smoother surface finish, while fewer flutes remove material fastest, but make rougher cuts.

Proper chipload is important because chips dissipate heat. Hot cutters can lead to suboptimal results, including burned wood, a poor edge finish and dull tooling.

If you’re machining a material like HDPE plastic, you want to use an “O” or single flute bit to clear the chips away as quickly as possible or heat will build up melting the plastic, which will “reweld” to the tool.

Single flute endmill

To summarize:

- More flutes create a smoother surface finish
- Fewer flutes are best at chip clearing, keep heat from building up
- Two or four flute cutters are the most common.

The direction, size, speed and amount of chips being ejected can also damage the surface of the work piece. We can control how the tooling effects the material through our end mill type selection (upcut, downcut or compression) and speed at which we cut.

Helical Direction, Chip Ejection, and Surfaces Produced

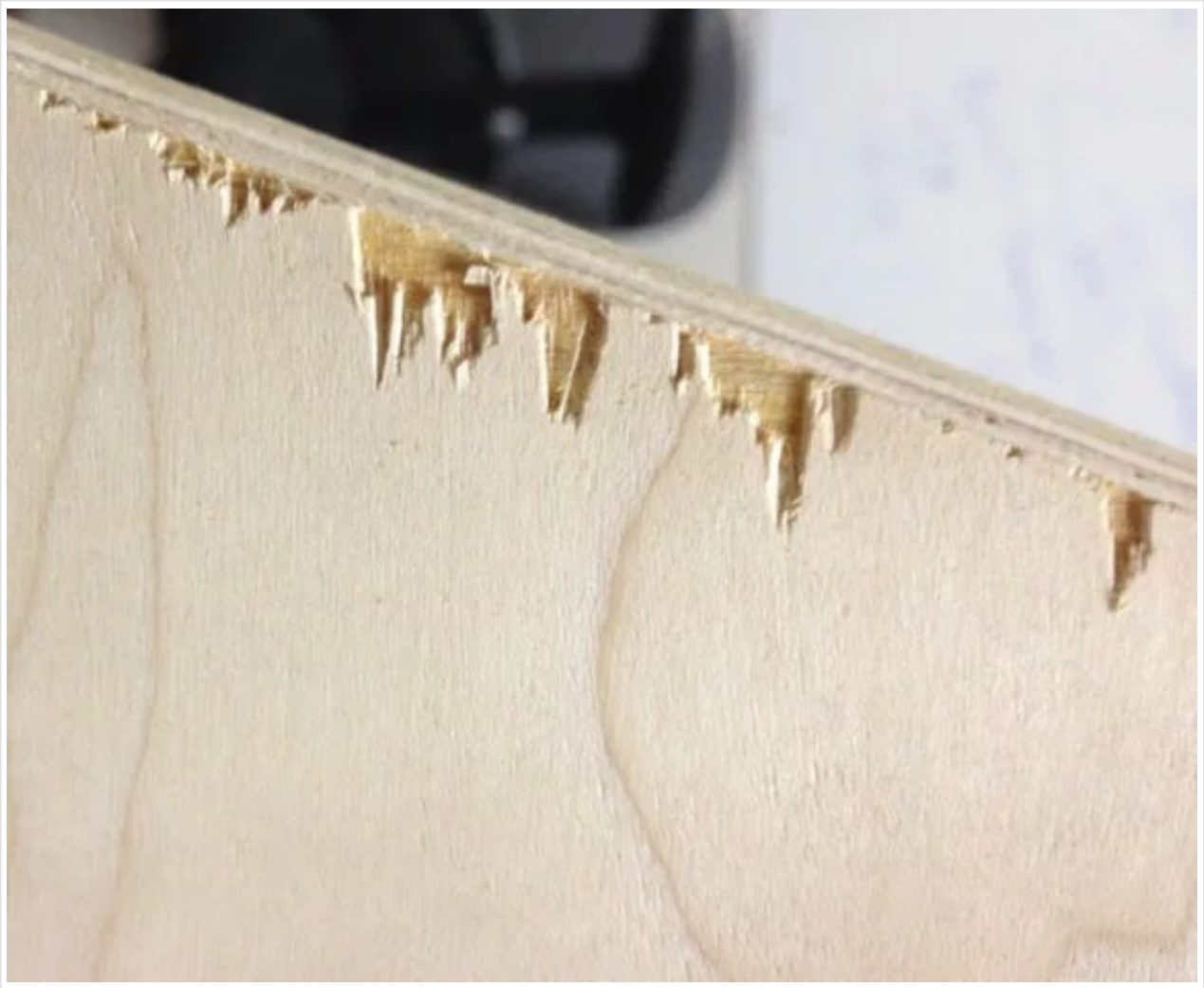
Downcut on left, upcut on right

A CNC router spins a cutter clockwise. The helical direction of the flutes as they wrap around the tool determine if chips are ejected towards the top or bottom of the workpiece.

True to their name, upcut mills eject chips towards the top of the workpiece, producing a cleanly cut bottom surface. The downside is possible surface splintering or “tearout” on the top surface as the chips are ejected upwards.

Downcut tools do the opposite, producing a smooth upper surface. They are ideal for pieces that have been previously engraved or v-carved and cannot be flipped

to hide tearout. In addition, as downcut mills pack the chips into the cut path, they can be used instead of tabs to hold down a workpiece and keep it from moving.



Plywood tearout caused by downcut chip ejection,

Compression end mills feed chips upward from the base and downward from the top side of the end mill. This type of mill produces a smooth surface on both sides when cutting through material. The sides of the cutting path however can be damaged by the chips. To

further reduce tearing in a wooden work piece, apply a layer of blue painters tape the surface. This will result in less surface splintering and can be easily removed when the milling operation finishes. If the desired finish is still not achieved, consider switching to wood stock that has a finer grain.

Feeds and Speeds

The speed at which we move a cutter across the material is called the “feed rate”. The rate of rotation is called the “speed” and is controlled by how fast the router or spindle turns the cutting tool. Both feed rate and spindle speed will vary based on the material being cut. A general rule of thumb is that you want to move the tool through the material as fast as possible, without sacrificing surface finish. The longer the tool rotates in any one place, the more heat that builds up. Heat is your enemy and can burn your material or radically decrease the life or your cutting tool. Feed rate vs spindle speed:

- Spindle speed that is too fast paired with a slow feed rate can result in burning or melting.
- Spindle speed that is too slow paired with a faster

feed rate can result in dulling of the cutting edge, deflection of the end mill and possibility of breaking the end mill.

A good strategy when selecting a cutter is to attempt to balance feed rate and spindle speed by performing two passes on the work piece. The first pass, called the roughing pass, can be done by using an end mill that will eject a large number of chips at a high feed rate. The second pass, called the finishing pass, then won't require as aggressive of a cut and can provide a smoother finish at a high speed.

Which Cutters to Buy First?

If you are looking to purchase a great wood and plastic starter set, consider picking up a few of the following carbide tool types in 1/4″ and 1/8″ diameters:

- 2 flute upcut and downcut end mills (great for hardwood and plywood)
- 2 or 4 flute ballnose mill (great for 3D contours)
- Single or “O” flute mill (great for plastics like HDPE and acrylic)
- 60° or 90° v-bit (great for cutting hardwood signs)

The quality of your work can be significantly improved by selecting the right tooling for your project and materials —plus you’ll spend less time on hand-finishing.

Original URL:
http://makezine.com/2014/09/10/endmills/