EML2322L – Design & Manufacturing Laboratory

Endmill Selection

1. **Select the cheapest tool that will do the job.** HSS (high speed steel) tools are approximately 2.5X cheaper than WC (tungsten carbide) tools.
2. **Select the toughest tool that will do the job.** HSS tools are much tougher (resistant to impact without chipping) than WC. Since HSS tools are also much cheaper, it's a proverbial win-win when machining nonferrous materials like aluminum.
3. **Select the largest/strongest tool that will do the job.** A ¼″ endmill is a lot stronger than an 1/8″ endmill, so unless absolutely necessary, try to select the largest tool that will do the job. The law of diminishing returns applies here, as once endmills reach ½″ in diameter, they are typically strong enough to cut anything we need to, and at that point larger tools just cost more money without much gain in strength / stiffness. Execution of this point often requires reevaluating the design to determine why a larger feature radius cannot be used to accommodate a larger cutting tool.
4. **Select the shortest tool that will do the job.** Almost every cutting tool used on a milling machine is essentially a cantilevered beam whose stiffness is inversely proportional to the cube of the length sticking out of the collet. So always select the smallest L:D (length-to-diameter) ratio possible for increased productivity, tool life, and surface finish. As a practical example, a tool that extends out of the tool holder or collet twice as much will be eight times more flexible.
5. **Select the appropriate number of flutes for the job.** Fewer flutes improve chip evacuation and more flutes improve tool stiffness and productivity (since more chips can be cut per each tool rotation). Do not use more than 3 flutes when full slotting in non-ferrous materials like aluminum, but use 4 or more flutes when cutting tougher materials like steels and titanium, or when making finish cuts in any material. Also, the higher the number of flutes, the larger and stiffer the endmill's core diameter.
6. **Understand the benefits of WC (tungsten carbide) tools (aka the 2.5 rules).** If you spend any time in the shop you will see tools made of WC, which in layman's terms has similar material properties to ceramics. WC tools can withstand approximately 2.5X more heat than HSS tool alloys (or more in the right application!). Coincidentally, WC is also about 2.5X stiffer than steel, which means it will deflect significantly less during heavy cutting. The downsides (as previously mentioned), are that WC is approximately 2.5X more expensive and much more brittle (less tough) than HSS, which is why both tool materials remain popular in modern manufacturing.
7. **Use roughing tools for roughing and save finishing tools for finishing.** Roughing tools are much stronger than finishing tools because they have generous fillets or chamfers on their cutting tips and serrated edges to break up chips into smaller pieces for improved evacuation and less chance of re-cutting. Using one tool to rough and finish wears it out much quicker, and often chips it before it even gets to the finish passes. So using roughing tools whenever possible can actually reduce the total tooling cost for the job.
8. **HSS or WC for finishing?** Because WC is made from a bunch of micro-grain powders, the cutting edge can only be ground so sharp. HSS can be honed to a sharper edge, but like an uber-sharp knife, it won't hold that sharper edge as long. So when trying to obtain the best surface finish possible cutting aluminum, HSS finish tools can actually work better, but they won't stay sharp as long. However, please do not interpret this point as saying you can't get a very nice finish with carbide in aluminum, because you most certainly can.
9. **Use the right tool coating for the job (or none at all).** The only tool coatings that work well when cutting aluminum are ZrN (zirconium nitride) or TiB_2 (titanium diboride). TiN (titanium nitride), TiAlN (titanium aluminum nitride), TiCN (titanium carbo nitride) are intended for cutting ferrous metals and tend to gall when cutting aluminum. Coatings typically allow for greater tool life or/or increased cutting speeds (between 10% and 25%).

10. **Select the proper helix angle.** Shallower helix angles provide stronger cutter edges for hardened materials, decreased axial forces and cutting aggressiveness, less potential for tool pull-out, less flute engagement and therefore less potential for chatter. Higher helix angles provide a greater shearing action and therefore lower power requirements, increased axial forces and cutting aggressiveness, higher potential for tool pull-out, and more flute engagement and therefore more potential for chatter.

11. **Understand commonly available endmill geometries.** Endmills are available with flat ends (the most common), ball ends, and convex radii in place of the normally sharp corners. Endmills are also available with concave corner radii for cutting fillets onto external corners.

12. **Use multiple tools when cutting deep features.** A standard length endmill may have flutes that measure 2×D in length, where D is the tool diameter. For example, a standard ½″ endmill may have 1″ of useable flute length. If cutting a feature that requires a longer endmill, always use a normal length tool first and only then switch to the longer tool(s) as necessary, since using the longer tool for the first inch of cutting depth would break Rule #4 above. In cases where the finish is important, longer endmills are also available with radially relieved shanks so they don't gall the portion of the part previously cut.

13. **A few cautions!**

    a. **Not all endmills are center-cutting**, meaning not all can be used to plunge mill (like a drill bit).

    b. **Endmills do not like to plunge**, as they have serious trouble with chip evacuation, which leads to chip recutting, and damaged cutting edges. Predrill a hole before plunging or ramp into the part using combined radial and axial displacement.

    c. **Damaged tools are still quite useful in forgiving materials, but not in tough materials.** Using a dull, damaged cutter in easy to machine materials like aluminum will simply result in a poor finish, which can be remedied by performing a finish pass with a nicer tool. So don't be so quick to grab the newest looking tool in the cabinet each time you have a part to make, especially if there's a lot of roughing to be performed. In addition, be encouraged to use worn or damaged tools to explore the limits of what they can do. BUT … do not try that with tougher to machine materials like stainless or titanium, as dull or damaged tools used in these materials will catastrophically overheat and fail before you have time to react.

    d. **Feeding an endmill too slowly is as bad for it as feeling it too quickly.** When the chip thickness becomes too small, each cutting edge is smearing rather than cutting, which produces significantly more heat and quickly dulls the cutting edge. The general rule of thumb is to not feed an endmill slower than 25% of its recommended feed per tooth. So if the suggested chip load for a ½″ endmill is 0.004″/tooth, bad things will start happening when you drop the feedrate lower than about 0.001″/tooth.

    e. **Cutter deeper produces proportionally higher axial forces.** The tangential cutting force on the endmill's helical cutting edge is equal to the cutting stiffness of the material times the chip thickness times the depth of cut. If you cut twice as deep, the forces are twice as large. This means you must be more careful to ensure the part is clamped securely when taking deeper axial cuts with the side of an endmill, even if only removing a small amount of material.