



END MILL SELECTION GUIDELINE

We often receive questions pertaining to properly selecting an end mill while programming a part. The following are great guidelines that could be helpful with those important questions/answers. Of course, we are always available to help assist you in selecting the correct product.

STEP 1 - MATERIAL

- > **Identify:** Know the exact material you are cutting, material condition (billet, forging etc.), and material hardness (HRC).
- > **Outcome:** This will guide you to our Non-Ferrous or Ferrous section of our catalog.

STEP 2 - OPERATION

- > **Identify:** Determine if you will be roughing, finishing, or both with this tool.
- > **Outcome:** This will help you determine the number of flutes needed and the need (or not) for chip breakers.

STEP 3 - PROGRAMMING

- > **Identify:** Will you be utilizing traditional programming, high efficiency programming (HEM), or a combination of both?
- > **Outcome:** This will help you decide on number of flutes in Step 8.

STEP 4 - ADOC (Axial Depth of Cut)

- > **Identify:** Determine the maximum axial depth of cut tool will experience in the part.
- > **Outcome:** This dimension (+1/2 x Dia. of tool) will help you determine the length of cut (LOC) to deploy.

STEP 5 - REACH

- > **Identify:** Do you have any obstacles to clear (fixturing, part features, etc.) or deep depths to reach (6x tool diameter or deeper)?
- > **Outcome:** May be time to employ a reduced necked tool to keep your length of cut reasonable but allow for you to drop the tool into a deeper position by stepping down (LOC max at a time) to a desired part depth.

STEP 6 - TOOL DIAMETER

- > **Identify:** Diameter selection takes a formulative consideration of programming type, machine taper, cut depth, reach, and part geometry (wall-to-wall corner requirements).
- > **Outcome:** Here are some helpful tips while deciding on and selecting the proper diameter.
 - If you're using a 40-taper machine, try and keep tool diameter <3/4" as this will extend your spindle life expectancy.
 - Determine what programming style (Step 3) you're using as HEM can employ smaller diameters than you may be used to.

- Decide on your cut depth (Step 4). For traditional programming keep it <2xDia., for HEM keep it below 4xDia.
- Decide on your total reach depth (Step 5). If needing to machine 4xDia. look at a necked tool to maintain strength and minimize deflection.
- Employ a tool radius (diameter/2) that is less than the part wall-to-wall radius requirement. For instance, for a .250-part corner radius constraint utilize a .375 or .437 Tool and generate the radius. Do not run a 1/2" end mill into a .250 corner it will chatter, squawk, and possibly break.

STEP 7 - CORNER RADIUS

- > **Identify:** Does your part require a corner radius specification to meet between wall and the floor?
- > **Outcome:** Running a corner radius on an end mill will increase its corner life. If your part requires one then great, if not, consider one for pre-finishing (aka roughing) as it will help to extend tool life!

STEP 8 - FLUTE COUNT

- > **Identify:** The cutting material (Step 1) and the type of programming in use (Step 3)
- > **Outcome:**
 - For Non-Ferrous machining (excl. Titanium)
Traditional Programming = 2-3 flutes, HEM = 3-5 Flutes
 - For Ferrous machining (incl. Titanium)
Traditional Programming = 4-5 flutes, HEM = 5-7 Flutes

Tip: the harder or tougher the material is, the greater number of flutes will help to increase tool life.

STEP 9 - TOOL HOLDER

- > **Identify:** Always try and use the most rigid and accurate tool holder proving the least amount of runout as possible. If you are planning to use a side lock holder, see the outcome below.
- > **Outcome:** It is best to keep TIR <.0005 at the tip of the tool (in the holder - in the machine) for optimum tool life and application success. Our tools come with cylindrical shanks. A Weldon Flat can be put on the tool by us quickly. Just order the tool by placing a "W" at the end of the EDP#. For example, EDP 60000 with a Weldon flat needed would be ordered as 60000W.

STEP 10 - Select & Pick the NEXGEN tool you need!

- > For **Roughing** tools go to pages **5-9**.
- > For **Non-ferrous** Tooling go to pages **11-15**.
- > For **Ferrous** tooling go to pages **17-25**.