

## Lathe Cutting Tool Selection

1. **There is a lot larger selection of cutting tools for turning than for milling.** Facing, turning, profiling, grooving (OD, face), parting, drilling, reaming, threading, knurling, etc.
2. **Select the toughest tool that will last long enough.** HSS tools are much tougher (resistant to impact without chipping) than WC. Additionally, WC is available in a variety of recipes that target applications requiring higher toughness or higher wear resistance, so choose wisely.
3. **Understand that while more brittle, WC, cermets, ceramics, or PCBNs may be the only choice when cutting higher strength materials that exceed the limits of traditional (high speed) steel based cutting tool materials.**
4. **In general there are three types of indexable toolholders: right hand, neutral, and left hand.**
5. **Learn how to navigate the alphabet soup of insert geometries (shapes, sizes, thicknesses, clearance angles, manufacturing tolerances, chipbreaker details, edge preparation details, etc.).** The [Engineers Black Book](#) and [the online guide at Carbide Depot](#) are two of the best references out there. Each tooling manufacturer has one in their catalog as well.
6. **Understand the three options for rake angle and when to use each.**
7. **Understand the sharper the angle of the selected insert, the weaker the geometry.**
8. **Select the proper carbide insert grade or prepare for disappointment.** Understand the correct insert grade (combination of toughness and hardness, edge prep, chipbreaker and coating) is imperative for decent tool life. The difference in proper grade selection can be 10 times the tool life or more!
9. **Use roughing tools for roughing and save finishing tools for finishing.** Roughing tools are much stronger than finishing tools because they have generous radii on their cutting tips and aggressive chipbreakers 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 tooling cost for the job.
10. **Select the shortest tool / toolholder.** Always select the toolholder with the shortest length that allows adequate tool clearance for the deepest depth cut and adequate nose clearance for anything with which it could collide (a part wall, vise jaw, clamping fixture, etc.). *Worded another way: always select the stiffest toolholder available that provides adequate working clearance.*
11. **A few cautions!**
  - a. **There is a minimum chip thickness below which cutting results are inconsistent.** Below a threshold thickness smearing becomes the dominant cutting regime instead of shearing.
  - b. **Feeding a cutting tool too slowly or with too shallow a depth of cut is as bad for it as feeling it too aggressively.** When chip thickness or depth of cut become too small, the cutting edge smears rather than cuts, producing significantly more heat and quickly dulling the cutting edge.
  - c. **Cutter deeper produces proportionally higher axial forces, whereas cutting faster does not.** The tangential cutting force on the endmill's helical cutting edge is equal to the cutting stiffness of the material times the feed per revolution 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 and the machine possesses adequate rigidity when taking deeper cuts or increasing the feed per revolution. However, increasing the surface speed has little effect on the cutting forces, which is why it's good practice to increase the depth of cut until chatter occurs, back off a little, and then increase surface speed until the desired insert life is achieved.