

Avoiding machining mistakes — before you turn the machine on

PLASTICS
MACHINING
AND EQUIPMENT

by Jack Sharp

Before you start milling, fly cutting, drilling, tapping, threading, sawing, boring, turning or parting off, one of the most important aspects of machining plastics is choosing the proper tooling. When Quadrant provides tooling recommendations, we typically suggest standard off-the-shelf tools that are readily available and easy to obtain.

Tool construction

The first consideration in tool selection is tool construction. Best results can be achieved with solid carbide because of its rigidity and long cutting life — although high speed cutters can be used on most materials. An exception exists for very abrasive materials where we recommend use of polycrystalline diamond or diamond coated tools.

Tool geometry

Next, we look at the tool's geometry. The ideal tool configuration is positive, but not overly positive. Many machinists believe that you need very positive geometries similar to those used on aluminum but these high positive geometries dig and pull at the plastic stock and may cause what looks like porosity or a poor surface finish.

For *milling*, Quadrant recommends using standard, two flute end mills that are uncoated. Our only use of coated tooling is for roughing cutters or taps. Uncoated tools are preferred, providing as sharp a

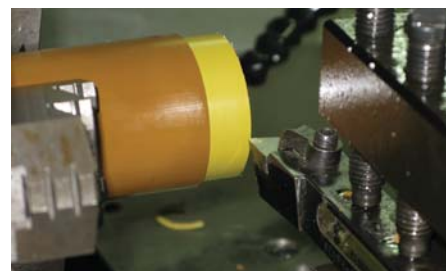
cutting edge as possible. When fly cutting or face milling, use inserted cutters with inserts that have a CPG or TPG geometry and holders that result in a zero degree to a slight positive lead angle. Do not use wiper inserts or inserts with chip breakers on them as the wiper will drag on the material, generating heat and causing a poor finish. To obtain a sharp cutting edge, use ground inserts and not molded or coated inserts. We also use inserts with a 30 thousand of an inch radius on the cutting point to help with achieving the best possible finish.

When *drilling*, use standard, uncoated twist drills. When drilling deeper than two and one half times the diameter of the drill, be sure to "peck" drill. Whenever possible, utilize inserted drills as they aid in chip removal and typically generate less heat and induce less stress into the material. An added benefit of using an inserted drill on a lathe is the chance to use it as a boring bar after rough drilling a hole. This eliminates a tool change and saves time. If drilling small diameter holes (less than .030" diameter) we strongly recommend use of solid carbide circuit board drills.

When *tapping*, use standard, straight two flute taps that are coated. Use of coated tools somewhat softens the cutting edge and adds a slight radius to the root of the thread being tapped. This helps to make the thread stronger. If you are chasing threads on a lathe here again, use a coated single point-threading tool to aid in putting a slight radius at the root of the thread for improved strength.

When *parting off* on the lathe, use a flat top part off tool and apply a 15-degree angle on the front to cut through the piece coming off prior to cutting all the way through the piece left in the lathe.

Sawing is where many people run into problems. This is primarily the result of incorrect blade choice. The most frequent error is use of a blade with too many teeth. Fewer teeth allow the chips an easier path out of the material and keep the chips from building up in the blade. Severe cases will cause heat to build up — possi-

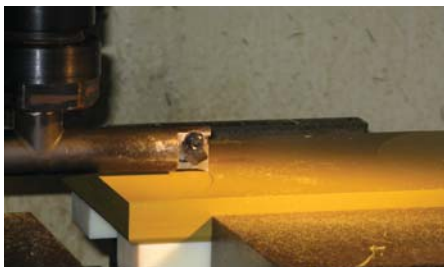


Turning Torlon® 4203 PAI from Quadrant is best done using uncoated, inserted tools.

bly to the point of melting the material and having the blade seize in the material. We recommend use of a three tooth-per-inch band saw blade with a nice set in the teeth so that the side of the blade does not "rub" the material and generate additional heat. Circular saw blades should be carbide tipped with a triple chip grind. If using a 14-inch diameter blade best success can be achieved with a 40 to 60 tooth blade unless sawing very thin plate where a 72-tooth blade is more appropriate.

Turning and boring are best done with inserted cutting tools with CPG tool geometry. Again, be careful not to use coated or molded inserts for these operations. Ideal results are achieved using ground inserts with a .030" radius on the cutting tip to yield a better finish. Chip breaker tools are not recommended as their molded construction does not adequately break the chips.

By choosing the proper tooling, you can be more aggressive when machining plastics resulting in shorter cycle times, improved dimensional control and optimal finish of the final part. Use of incorrect tooling can induce stress into the plastic material, cause dimensional problems and keep you from achieving the best finish. ■



Milling operations, like the fly cutting of Quadrant's Torlon® 4203 PAI should be done with cutters of the TPG or CPG variety. Details on how to achieve the best tolerance and finish can be found on Quadrant's web site at www.quadrantep.com.

Jack Sharp is machine shop/tooling manager for Quadrant EPP USA, Inc. For further information, contact Quadrant EPP USA, Inc., P.O. Box 14235, Reading, PA 19605 USA; (610) 320-6600, (800) 366-0300, fax (800) 366-0301, e-mail: americas.epp@qplas.com, www.quadrantep.com.