Machining Polyurethanes: Turning O.D.'s & Facing

Turning, boring, and facing operations can be performed on either a turret or engine lathe. Tool configuration, geometry, and placement, as well as rpm are dependent on what hardness the urethane is and what the operation is. Another variable is feed rate – the speed of the tool with respect to the rpm of the lathe. Feed is often controlled by hand and is subject to operator judgement and "feel".

In general, use sharp tools, high turning speed, and slow to moderate feeds (depending on hardness). Cutting tools for urethane must have sharp, carefully honed cutting edges. Sharpen tools on a honing stone for a razor sharp edge on the sides, tip, and top of your tool. We have found success with both high speed steel and carbide tools.

Tool clearances must be greater than those used for metal. The goal is to have little or no resistance as the tool travels through the urethane. The chip (material that is being cut away) should come off as a continuous strip or ribbon. A smooth surface on the top of your tool will aid in chip removal. This is very important to prevent the chip from wrapping back around the workpiece. Good chip removal is also critical for heat removal and tool life. See figure 1.

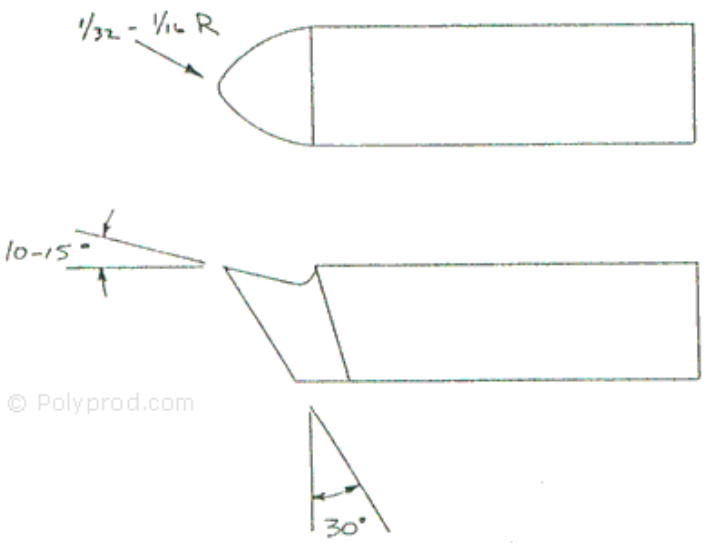

Figure 1. TURNING / FACING TOOL

Tool geometry is very important for successful machining of urethane. Rake angles and nose radius will vary depending on durometer and desired surface finish. TOP RAKE determines chip flow. Too little causes that build-up and poor chip removal. Too much will cause reduced tool life. 10 to 15 degrees is a good starting point. SIDE RAKE is the amount of angle from the cutting edge to the bottom of the cutting tool 30 degrees is a good starting point for side rake. This angle affects surface finish. Too little allows the tool to rub against the workpiece. Too much will shorten tool life. NOSE RADIUS is the radius of the top edge of the tool at the tip. Nose radius is the most important part on a urethane tool because it significantly affects surface finish. As a rule, as durometer goes up, nose radius increases.

High durometer urethanes (95A and up) can be turned very easily. Smooth surfaces can be achieved on heavy as well as light cuts, so roughing cuts are seldom required. Surface finish is best when your removal is .050" or more so your cutting tool gets a good "bite". We have found that on the harder urethanes, a speed of 600 to 1000 rpm works well. Feed rate depends on the desired surface finish. Typical feeds are in the .005 to .010 in./sec, range. The faster the feed rate, the more of a "record effect" you will have in your surface finish.

For higher durometer urethanes, a round-nose tool is a good choice. The radius on the cutting end affects surface finish. The nose radius should be about 1/16 with a 10 to 15 degree top rake and a 30 degree rake around both sides.

Medium hardness urethanes - 80A to 90A - require a tool with a smaller nose radius of 1/32 or less. Side and top rake should stay about the same. RPM should be faster than what is used for harder durometers - in the range of 1000 to 1500 rpm. Feeds are generally faster - in the range of .050 to .150 in./sec. This is a rapid "plunge" type feed.

When turning large diameter parts, cuts of 1/10 to 1/8 inch deep and a light feed of .003 to .007 inches per revolution are recommended. Remember also that centrifugal forces on large, low hardness parts can cause a "flaring" type deformation at high turning speeds. This effect can cause the part to be machined improperly if it is not compensated for.