Guide to Machining Cast Polyurethane Elastomers

Cast polyurethane can be readily turned, sawed, drilled, ground or milled. These and other secondary operations present many similarities to the machining of metal, but there are also some important differences. This following information is intended to provide some general guidelines for machining urethanes.

It is important to note that the material presented here is a starting point. The wide variety of urethane compounds and their respective physical properties and characteristics create a wide range of machining situations.

Harder urethanes – 90 Shore A and up – have a high degree of machinability. Lathe turning, flycutting, grinding, contouring and milling are more easily accomplished on conventional metalworking equipment by machinists who are familiar with procedures for handling plastics.

Some different tools and techniques are required for compounds of 80 Shore A durometer and lower. These lower modulus compounds are typically machined by knifing, grinding, and sanding. In some cases, however, they can be worked like higher modulus materials by “freezing” them in dry ice or liquid nitrogen environments.

Ultimate dimensions can be affected by temperature and humidity. Polyurethane will show small changes in dimensions when subjected to high humidity and/or high temperatures. Machined parts should be stored at ambient temperatures and 50% relative humidity.

Sawing

One of the best machines for sawing urethane is a band saw. Long blades (125 to 175 inches) run cooler and help prevent the urethane from melting. A hook type tooth blade approximately ½” wide by .030” thick having four teeth per inch has been found to work well. Surface speeds of 200 ft./min. work well on all hardnesses.

For a finer finish use a 10 tooth per inch blade. When cutting 90 Shore A and below, the use a spray mist of water-soluble oil helps to keep heat down and improve the finish.

Turning, Facing, & Boring

Turning, facing, and boring operations are performed on either a turret or engine lathe. Tool configuration, geometry, placement, and speed are dependent on the hardness of the urethane and the operation selected.

Nominal Turning Conditions:

Durometer or Hardness	Cutting Speed ft/min	Feed in./rev.	Shape of tool	Surface Roughness, Microinches
78A to 88A	1,000 – 1,650	.004 to .008	12 53 25	50
89A to 95A	330 – 500	.004 to .008	12 53 25	20
50D to 60D	330 – 500	.004 to .008	12 53 25	10

Facing

The conditions for facing board parts are the same as for turning. When thin discs are to be cut, use a cutting tool with a very acute blade (approximately 15 degrees included angle). Owing to the heat development resulting from friction, cooling should be incorporated.

Boring

Normal tools are used at cutting speeds of 130 – 170 ft./min. and as slow a feed as possible (approximately .0004 to .0012 in. /min.). A faster speed can be employed with harder materials. Hole depths up to one inch can normally be made without the use of coolant. Hole diameter will be up to 4% less than the diameter of the drill for materials less than 80 Shore A.

Knifing

Knife cutting urethane to close tolerance should be done with a razor sharp tool, and the tool must be as thin as possible.

Durometer or Hardness	Tool	Turning Speed RPM	Feed Rate
70A to 95A	High Speed Steel	600 to 1,000	Rapid, hand
Below 70A	Ground carbide	600 to 700	Moderate to rapid, hand

Milling

Urethanes ranging from 90 Shore A to 75 Shore D can be milled successfully. However, attempting to parts less than 80 Shore A is not recommended.

Tool	Speed	Feed Rate
- Two fluted end mills - Single point fly cutter	900 to 1,300	15 to 20 inches/min.

Drilling

Slow spiral drills are recommended. This permits free discharge of chips with minimum binding and heat buildup.

Sharp cutting edges are a must. Polished flutes are required to aid in chip clearance, and coolant is required for good drilling performance.

The rake angle should be reduced to 0-degree angle, and a generous lip clearance (approximately 16 degrees) provided for proper relief.

Sharp points of 90 – 110 degrees are best for heavy walls and large diameters, while blunt angles (115 to 130 degrees) are better for thin walls. Close tolerances require feeds of .004 to .006 ipr.

Grinding

Smooth surfaces are obtained with grinding wheels of carborundum with a fine grain size, medium hardness, and coarse texture. A 6” x ¾” Simonds wheel C60-J-7B-3 works well. RPM of the wheel should be in the range of 2,200 to 3,500.