CAST NYLONS LIMITED
4300 Hamann Parkway, Willoughby, Ohio 44094
800-543-3619 • www.castnylon.com

**Technical Bulletin #106**

# MACHINING

## NYCAST MACHINING INSTRUCTIONS

The following machining instructions are applicable for all NYCAST materials:

***Turning*** NYCAST materials should be turned the same as a free-cutting material, using high speeds of 600 to 900 feet per minute surface speed, with heavy roughing cuts at feed rates per revolution of 0.004 to 0.010 inches (0.010 to 0.25m) for smaller diameters, and 0.003 to 0.007 inches (0.08 to 0.18mm) for larger diameters.

In turning large diameter pieces of NYCAST materials, light cuts of 1/16 to 1/8 inch deep and light feeds of .003 to .007 inches per revolution are recommended for the final pass. Satisfactory turning results can be achieved, however, using roughing cuts up to 3/8 inches deep and feeds of .015 inches per revolution.

Sharp, honed tools with low rake and high clearance angles are suggested to minimize cutting forces, reduce heat build-up, and obtain the best results when turning, boring, facing, or milling. Tools made of high speed tool steel are generally adequate for turning NYCAST materials, carbide-tipped tools may be used for longer production runs if the tools are honed to a very sharp edge.

***Milling*** Cutter speeds in excess of 1000 feet/minute (5m/second) with fast feed rates of nine feet/minute (4mm/second) and heavy cuts are commonly used for NYCAST materials. Milling cutters designed for light metals can be used, but fly cutters generally perform better because of swarf removal. Milling cutters must be sharp and have high positive cutting angles.

For milling operations, the workpiece must be fully supported during all operations on the mill. When clamping or holding is required, it is important to exercise care to prevent deformation of the workpiece.

***Sawing*** All NYCAST materials can be sawed on standard woodworking or metalworking band saws, and circular saws. The blades should have widely spaced teeth to assure adequate chip removal. In order to avoid excessive heat build-up and possible binding of the saw, blades should have enough set to accommodate the tendency of nylons to close-in behind the cutting edge of the blade.

***Drilling*** Drilling is the most difficult of all nylon machining operations due to the confined space in which drills operate and the poor heat conduction of nylons. Proper tooling and procedures, however, will eliminate problems such as gumming, melted hole surface, cracking, and possible part failure.

Although properly ground, standard twist drills can be used satisfactorily, slow spiral drills with their larger flute areas provide a clearer path for chip flow. For best results, use a new drill and grind the tip to thin the web area and provide a 0-5 degree positive rake angle at the cutting lip.

## Standard Twist Drill Terminology

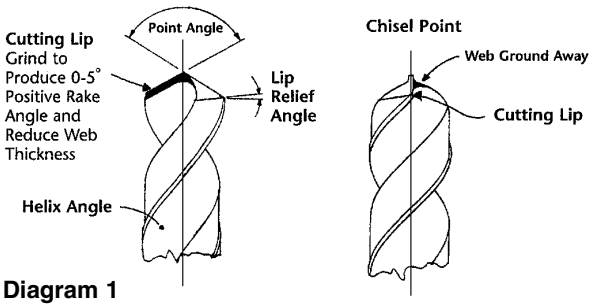


**Diagram 1**

A drill point angle of 90-110 degrees is best for small (under 1/2") drills, while a point angle of 118-120 degrees is better for larger (over 1/2") drills. All drills should have a lip relief angle of 10 to 15 degrees. Standard drills previously used for metals should never be used for nylons. Use of coolants, such as soluble-oils or mist spray, together with frequent drill pull-out (peck drilling) are essential to successful drilling operations. A good guide for peck drilling is to pull the drill out of the hole after drilling to a depth not more than 1 - 1/2 times the drill diameter. When drilling large or deep holes, start with a small (maximum 1/2" diameter) hole drilled at a speed of 800 to 900 rpm and

# MACHINING cont.

a feed rate of .005" per rev. The web area and cutting lip must be ground as in diagram 1 to prevent "grabbing" and stress cracking. Open the hole to 1" following the same procedures but using a drill speed of 400-500 rpm. ***Peck drill and use generous amounts of coolant for each operation***.

To open the hole to finished size, use a single point boring tool and follow the procedures in the "turning" section.

***Reaming*** Whenever possible, reamers of the expansion type should be used, and reamer speeds should approximate those used for drilling (250 to 450 feet per minute). Feed rates should be between 10 and 20 mils per revolution. Since it is difficult to remove less than .002 inches when reaming, it is best to leave at least .005 inches for final reaming. This will provide a "bite" for the reamer and will assure accurate cutting.

## Feed Rate Up To 1" Dia. Drills

| Feed Rates/rev | 1/16" dia. | 1/4" dia. | 1/2" dia. | 1" dia. |
|---|---|---|---|---|
| 0.004" – 0.015" | 5000 rpm<br>1500 rpm | 1700 rpm | 1000 rpm | 500 rpm |
| 0.008" – 0.016" | 3500 rpm | 1500 rpm | 1000 rpm | 500 rpm |
| 0.013" | 3000 rpm | 1000 rpm | 750 rpm | 400 rpm |

## Drill Speed/ Size Chart

| Drill Sizes | RPM |
|---|---|
| No. 60 thru 33 | 5,000 |
| No. 32 thru 17 | 3,000 |
| No. 16 thru 1 | 2,500 |
| 1/16" | 5,000 |
| 1/8" | 3,000 |
| 3/16" | 2,500 |
| 1/4" | 1,700 |
| 5/16" | 1,700 |
| 3/8" | 1,300 |
| 7/16" | 1,000 |
| 1/2" | 1,000 |
| A thru D | 2,500 |
| E thru M | 1,700 |
| N thru Z | 1,300 |

***Tapping*** The tapping of NYCAST materials can be performed either by hand or by machine; however, the use of sharp taps is essential. Taps previously used on metal should never be used on nylon workpieces. In tapping, high speed oversize taps, such as H-3 oversize, can be used for smaller diameters and H-5 oversize for larger diameters. Any high speed tap used should be oversized by 0.002 to 0.005 inches (0.05 to 0.13mm).

***Threading*** As in tapping, dies must be sharp and should never have been used on metal. Threads can be cut with any conventional method, but dies must be well backed-off to avoid non-cutting surface contact with the workpiece. Threads may be cut with a single point tool. Light cuts of less than 0.005 inches should be avoided, and a maximum cut of 0.010 inches is suggested. Heavy cuts may be used on the initial pass, but the depth of the cut should be reduced to 0.007 inches on the final pass. Since nylon materials have a tendency toward memory or recovery after the die is removed, a slightly oversize die should be used for threading (see tapping section).