CAST NYLONS LIMITED

Discover the NYCAST® Advantage

Machining Guidelines

Distributed by: Polymershapes

Machining Guidelines

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 1000 surface feet per minute, with heavy roughing cuts at feed rates of 0.010″ to 0.020″ (0.025 mm to 0.50 mm) per revolution, for large diameters, and 0.003″ to 0.007″ (0.08 to 0.18mm) for smaller diameters, due to material deflection and material expansion during the material removal process. In turning large diameter pieces of NYCAST materials, light cuts of 1/16″ to 1/8″ inch deep and light feeds of 0.003 to 0.007 inches per revolution are recommended for the final pass, recommending that if possible the finish pass is made with a 24 hour relax period if possible. Satisfactory turning results can be achieved, however, using roughing cuts up to 3/8″ deep and feeds of 0.015″ per revolution. Sharp, honed tools with high rake and high clearance angles are suggested to minimize cutting forces, turning, boring, facing, or milling. Tools made of high-speed tool steel are generally adequate for turning NYCAST materials but carbide-tipped tools may be used for longer production runs, if the tools are honed to a very sharp edge (<.010″).

MILLING

Cutter speeds of 1000-4000 surface feet per minute, with fast feed rates of 0.010″ to 0.020″ per cutter tooth combined with heavy depths of cut are commonly used on NYCAST materials. Indexable insert milling cutters designed for nonferrous metals are general recommended, but fly cutters can be used and perform well but are significantly slower due to material removal rate. Milling cutters must be sharp and have high positive cutting angles and high rake angles. For milling operations, the work piece 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 work piece.

SAWING

All NYCAST materials can be sawed on standard woodworking or metalworking band or circular saws. The blades of both should have widely spaced teeth that are offset (left, right, and center) and have a deep tooth gullet. This will ensure the tooth of the blade stick out farther than the saw body and gives the chips a place to sit while being forced through the material. Evacuation of the chip immediately after the saw has exited the material will help to avoid pinching/binding of the blade in the material due to material heat buildup. This is commonly done through the use of an air blast or semi-synthetic coolant hose.

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/fracture. 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.

Machining Guidelines (continued)

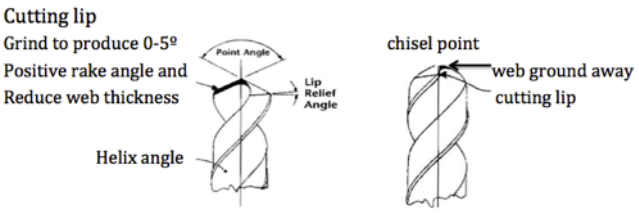

Figure: standard twist drill terminology

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. The use of coolants, such as semi-synthetics in flood or mist spray, together with frequent drill pullout (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 1/2 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 a feed rate of 0.005″ 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 0.002 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 rates/ rev.	1/16 ″dia.	¼″dia.	½″dia.	1″dia.
0.004″-0.015″	5,000 rpm 1,500 rpm	1,700 rpm	1,000 rpm	500 rpm
0.008″-0.016	3,500 rpm	1,500 rpm	1,000 rpm	500 rpm
0.013	3,000 rpm	1,000 rpm	750 rpm	400 rpm

Chart: feed rate up to 1″dia. drills

Drill size	RPM
No. 60 thru 33 No. 32 thru 17 No. 16 thru 1	5,000 3,000 2,500
1/16″ 1/8″ 3/16″ ¼″ 5/16″ 3/8″ 7/16″ ½″	5,000 3,000 2,500 1,700 1,700 1,300 1,000 1,000
A thru D E thru M N thru Z	2,500 1,700 1,300

Chart: drill speed/ size chart

Machining Guidelines

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 work pieces. 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 work piece.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)

POST MACHINE ANNEALING

All material grades from Cast Nylons are annealed after processed to reduce internal stress, which may have resulted from the manufacturing process. The annealing process guarantees the dimensional stability of machined parts made from stock shapes.

Some parts might require intermediate annealing in between machining operations. such as:

- Parts were large volume is machined away, especially from one side

- Special tolerances or flatness needed

Other parts might need post-annealing process after finished machined if:

- Machined with dull tools

- Machined without cooling and created excessive heat

The annealing process should be done in an oven under nitrogen atmosphere or in an oil bath. The heat up rate should not exceed 150°F/ 10 minutes. Annealing temperature to be set at 280° to 300°F. Annealing time should be 30 minutes per ¼" thickness of the part. The cooling rate should not exceed 300°F/ hour.

For more information, please contact CAST NYLONS LIMITED.

CAST NYLONS LIMITED

Certified to ISO 9001:2008

member iapd

Distributed by: Polymershapes

Call: 1 (866) 437-7427
Email: info@polymershapes.com
www.polymershapes.com