General Information

Cast Nylon Rods, Tubes and Plates are produced almost entirely for the purpose of providing stock for subsequent manufacture of machined components.

The range of NYLATECH materials all machine easily, and in particular the NYLATECH OIL and NYLATECH SL grades due to the proprietary lubricants impregnated during manufacture. Virtually all types of woodworking and metalworking equipment can successfully be employed in machine finishing applications. However, some minor modifications to standard metal/wood tooling must be made to accommodate the unique properties of Cast Nylon.

Machining tools should be kept extremely sharp, and have sufficient clearance so that only the cutting face comes in contact with the material.

Frictional Heat:

Any heat generated by the friction of the tool upon the surface of the nylon will effect the ultimate surface of your finished part. Most frictional heat can be avoided by proper tool design and feed speeds, or absorbed by the tooling. In precision turning, were thermal expansion will effect the tolerances, it is recommended that a coolant be used to aid heat dissipation. Acceptable coolants are Compressed Air, mist sprays, soluble-oil, or light cutting oils. Frictional heat will cause nylon to expand. For tight tolerances and/or proper interference fit, it is recommended to machine to within one pass of the final dimension and allow the part to sit overnight, then machine the final pass.

Resiliency

Nylon is very resilient when compared to metals, and special care must be given when it is supported to prevent distortion. A 4 or 6 jaw chuck is recommended when turning, and for thin parts (wall thickness' less than 1/2") soft jaws are recommended to adequately support the entire part.

Moisture Absorption:

Cast Nylon is a hydroscopic material, which absorbs water both from humidity, and submersion. Before machining precision parts, it may be necessary to consider the moisture factor before machining. One of two possible methods may be used; First, the moisture expansion percentage can be used to calculate the expected amount of growth due to moisture. This can then be subtracted from your finished dimensions before machining. Second, the material can be conditioned by submersion in water prior to machining. Material should be submerged for at least 48 hrs @ 73° F.

Annealing

Post Machining annealing has proven effective for reducing stress created during machining. Causes of this stress include: using dull tooling, using tooling inappropriate for machining nylon, excessive heat generated from improper speeds and feed rates, and machining large amounts of material from one side of a stock shape. For information, please see the machining guidelines below.

To reduce this stress annealing should be done just prior to the finished dimensions are obtained. The annealing process is as follows:

1. Heat the nylon up to 300°F slowly. This should take at least 4 hours.
2. Hold the material at 300° F for ½ hour per ¼" of thickness.
3. Cool the material down at a rate of 30° F per hour until the material has reached room temperature.

When annealing strips, the material must be fastened to a straight anchor to eliminate any possibility of bowing.

Precision Turning

Carbide steel tools ground to the following specifications are recommended for most turning applications. HS steel tools are acceptable for non-critical operations, providing they are regularly sharpened. Special attention should be paid to the negative Rake Angle to prevent the material from being pulled into the cutting tool. As well, ample end relief angles will prevent excess heat build-up in the tool tip.

Cutting Speed	330 - 3200	fpm
Feed	0.002 - 0.020	ipr
Rake Angle	0-10 (Negative)	Degrees
Side Relief Angle	5 - 15	Degrees
End Relief	5 - 40	Degrees
Tip Radius	0.02 - 0.065	inch

Drilling

Any commercial grade drill for metals is suitable for use with NYLATECH Cast Nylon. However, due to the extreme frictional heat generated during drilling, care must be taken to observe the following conditions.

Drill Speed	$\leq$ 350	fpm
Feed	.004" - .015"	in/rev
Rake Angle	Negative 5 to 0	Degrees
Helix Angle	14 - 16	Degrees
Point Angle	90 - 118	Degrees
Lip Relief Angle	9 - 16	Degrees

A short drill best allows holes of consistent dimensions to be drilled in one operation. To prevent chip buildup, the drill should be frequently removed from the hole. A guideline is to never allow the drill to exceed 2 - 3 times the drill diameter before withdrawal. Coolant is recommended to aid heat dissipation; preferred coolants are compressed air, soluble-oil, or light cutting oil.

When drilling through holes, the feed should be reduced as the tip reaches the end of the cut to prevent the drill from pulling through, breaking out or chipping.

Special care should be taken when boring large diameter holes to prevent stress building up, or material “Grab” and cracking. A small pilot hole (1/2” to 5/8”) should be made at 600 to 1000 rpm using a positive feed of .005”/rev. Once the pilot hole has been bored, step up in 0.75” increments until desired ID is reached. Always use an automatic feed to prevent material from binding on the flute.

Milling

Nylon castings can be milled by conventional means, e.g., with shank - type cutters, hobbers, and face cutters. Fly cutters are preferred because of their superior rate of swarf removal. High spindle speeds, and fast table travel are best utilized under the following conditions:

Cutting Speed	< 3500	fpm
Feed	< 0.020	ipr
Rake Angle	15 - 20	Degrees
Relief Angle	8 - 10	Degrees
Lip Angle	25 - 35	Degrees
Cut Depth	0.075 (Smoothing) 0.200 (Roughing)	inch

Due to the resiliency and elasticity of Cast Nylon parts, special attention should be given when holding and clamping parts to prevent deformation. Thin wall parts should be enclosed within the clamping structure, leaving only enough room for the cutting bit, this prevents deflection. Some components such as wear strips can be held to the bed using double sided adhesive tape.

Coolant is rarely required for milling applications, as it impedes chip removal. If overheating is a concern, compressed air should be directed at the cutting bit.

Sawing

The use of triple-tip table saw blades or off-set band saw blades yield very clean cuts under the following conditions:

Circular Saw

Cutting Speed	3,000 - 10,000	fpm
Rake Angle	0 - 10	Degrees
Relief Angle	10 - 15	Degrees
Pitch	0.313 - 1.750	inch

Band Saw

Cutting Speed	600 - 3,000	fpm
Rake Angle	0 - 8	Degrees
Relief Angle	30 - 40	Degrees
Pitch	0.125 - 0.500	inch

Under an adverse combination of circumstances, there is a risk that the cut surface of nylon castings can melt during sawing. This is due to the relatively high cutting speed in relation to the slow feed speed of the material. To counter this effect, a vacuum system should be employed to prevent chip from building up around the blade. As well, coolants can be used in the form of blade lubricants or compressed air.

When cutting thicker material, there is a tendency for the material to “close” behind the blade. Blades should have enough kerf to allow proper clearance. Sometimes a wedge can be used to hold the material open. Although, when employing the use of a wedge, care must be taken to prevent the cut from widening beyond the saw kerf or the material may split through.

Feed speeds vary depending upon depth of the cut and blade configuration. Keep a constant steady pressure that allows the chip to clear from the blade area. Too aggressive feed pressure will bind the blade, and possibly stall the saw, where too light a feed pressure will overheat the blade and cause a gummy, melted surface finish.

Planing

NYLATECH Cast Nylon stock sheets can be planed to thickness with acceptable surface finishes, and tolerances can be held from .025" to .005". Under the conditions listed below, coolant is not required, or recommended. Cutting speeds are not limited by the work-piece, but are usually the highest that can be obtained on commercial planers for wood and metal.

Feed		0.012 - 0.032	inch/stroke
Rake Angle		25 - 40	Degrees
Relief Angle		8 - 10	Degrees
Radius of Curvature		$\geq$ .0393	inch
Cut		$\leq$ 0.25	inch

At feeds greater than 0.012" per stroke there is a risk that material may break off. This risk is increased when planing material $\leq$ 0.5" thick, or when planing 6/12 material.

Threading

Threads are cut in nylon in very much the same way as metal; external/internal threads with a single-point threading tool or tap and external threads with a die.

British Standard series, or American Standard Uniform thread form (with rounded root) is recommended when threading nylon. This will prevent thread failure due to the "notch" effect.

Four Flute taps are preferred to aid chip clearance, and prevent heat buildup. When tapping deep holes, greater than 3:1, the flute area should be enlarged for greater chip clearance.

Single Point Threading Tool

Speed	300 - 500	rpm
Cutting Speed	100 - 320	fpm
Depth of Cut	0.0078 - 0.0315	inch

Die

Speed	100	rpm
Cutting Speed	20	fpm

Tap

Speed	600	rpm
Cutting Speed	125	fpm

A conventional drilling emulsion is recommended for cutting threads with a tap. Due to the elasticity of the nylon, the threads may be too tight. This may be overcome using a chrome-plated or oversized tap.