

## Group 2: Unipa<sup>®</sup> Nylon Products

### GENERAL NOTES FOR MACHINING NON-REINFORCED UNIPA EXTENDED NYLON AND CAST NYLON PRODUCTS:

1. Compressed air or water soluble liquid coolants are recommended when drilling, reaming, tapping, threading, or cutting very thick materials.
2. Unipa Nylon is not abrasive. Most machining can be efficiently completed with High Speed Steel (HSS) cutters and tooling. On very large volume production jobs, Carbide tooling may be used during some operations to further reduce the amount of re-sharpening that is required.
3. Although Unipa Nylon is not difficult to machine, achieving very high quality surfaces may require experimentation with tools, feeds and speeds. Very sharp HSS tools are required for high quality surfaces.
4. Nylon products tend to create burrs during machining. While sharp tools will minimize burrs, trimming by hand is usually required to remove all of them.
5. Nylon products will slowly absorb moisture when they come in contact with liquids and also directly from humid air. This absorbed moisture will cause the nylon to swell. Under dry environments, Nylon will lose moisture and shrink. This swelling and shrinking must be taken into account when machining tolerances are developed.

### BAND SAWING

Material Thickness	Pitch (teeth/in.)	Speed (sfpm)
<1"	6 - 8	3000
1" - 3"	4 - 5	2000
3" - 4"	2 - 3	1000

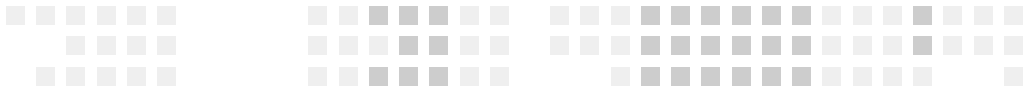
### CIRCULAR SAWING

Material Thickness	Pitch (teeth/in.)	Speed (sfpm)
<1"	2	9000
1" - 3"	1	9000
3" - 4"	1	9000

#### NOTES:

- |                      |           |               |
|----------------------|-----------|---------------|
| 1. Tooth Geometries: | Band Saws | Circular Saws |
| Rake Angle:          | 2°- 8°    | 2°-10°        |
| Clearance Angle:     | 30°- 40°  | 20°- 30°      |
2. To prevent binding, saws should have a slight set (3°-10°). Coarse teeth with wide gullets are recommended.
  3. Larger diameter circular blades and thicker band saw blades promote cooling and reduce heat buildup.
  4. Thinner stock requires more teeth per inch.





## TURNING/THREADING

Operation	Tool Material	Depth of Cut	Speed (sfpm)	Feed (in./rev.)
Rough Cutting	Carbide	0.150"	500 - 800	0.005 - 0.020
General Turning	Carbide	0.100"	800	0.010
Finish Cut	HSS/Carbide	0.025"	800 - 1000	0.002 - 0.005
Cut Off	HSS/Carbide		700	0.004 - 0.015
Threading	Carbide		500	0.003 - 0.005

### NOTES:

1. Tooth form recommendations:

- Top Rake Angle: 0°-5°
- Clearance Angle: 5°-15°
- Side Incidence Angle: 30°-60°

2. Carbide inserts are adequate for most turning operations. For operations requiring high quality surface finishes, High Speed steel tools are recommended because of the sharper cutting edge that can be held on the tool.

3. When threading, use a single point and finish with several 0.001" passes. The use of coolant during threading is recommended.

## MILLING

Operation	Tool Material	Depth of Cut	Speed (sfpm)	Feed (in./rev.)
Face Milling - Rough	HSS/Carbide	0.150"	1000 - 1500	0.020
Face Milling - Finish	HSS/Carbide	0.050"	2000 - 2500	0.005
End Milling - Rough	HSS/Carbide	0.250"	250 - 450	0.002 - 0.010
End Milling - Finish	HSS/Carbide	0.050"	350 - 550	0.001 - 0.005

### NOTES:

1. Tooth form recommendations:

- Rake Angle: 0°-15°
- Clearance Angle: 5°-15°

2. Single or dual fluted cutters are desirable because they produce less heat and vibration than multi-fluted helical cutters.

## DRILLING/REAMING

Operation	Tool Material	Depth of Cut	Speed (sfpm)	Feed (in./rev.)
Drilling	<1" dia.	HSS/Carbide	150 - 450	0.004 - 0.015
Drilling	≥1" dia.	HSS/Carbide	150 - 250	0.010 - 0.050
Reaming		HSS/Carbide	300 - 450	0.005 - 0.015

### NOTES:

1. During drilling and reaming, assure that chips do not build up in the hole. Failure to adequately clear chips will cause melting, cracking, and oversize holes. Peck drilling is recommended.

2. For small holes (<1" dia.) high speed steel twist drills are sufficient.

3. For larger holes (≥1" dia.), use a low helix bit with a point angle of 90°-118° with a relief lip clearance of 10°-15° and 0°-5° rake angle. To minimize cracking, a 1/2" pilot hole should be drilled prior to finish drilling to the required size.

4. During reaming, use a 0.005 - 0.010" depth of cut. To avoid undersized holes, the final cut with a fixed reamer should be at least 0.005". Helical flute reamers are recommended if there is an interruption in the I.D. of the hole.



**NYTEF** PLASTICS

**800-646-9833**

sales@nytef.com