

Group 4: Reinforced Amorphous High Performance Materials

GENERAL NOTES FOR MACHINING NON-REINFORCED AMORPHOUS HPM'S:

1. Compressed air or water soluble liquid coolants are recommended when drilling, reaming, tapping, threading, or cutting very thick materials.
2. Most amorphous stock shape plastics are 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 these materials are 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.

BAND SAWING

Material Thickness	Pitch (teeth/in.)	Speed (sfpm)
<1"	6 - 14	4000
1" - 3"	4 - 5	3000
3" - 4"	2 - 3	2000

CIRCULAR SAWING

Material Thickness	Pitch (teeth/in.)	Speed (sfpm)
<1"	6 - 12	9000
1" - 3"	2 - 4	9000
3" - 4"	2 - 4	9000

NOTES:

- | | | |
|----------------------|-----------|---------------|
| 1. Tooth Geometries: | Band Saws | Circular Saws |
| Rake Angle: | 0°- 8° | 0°-10° |
| Clearance Angle: | 15°-30° | 15°-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"	800 - 1000	0.005 - 0.015
General Turning	Carbide	0.100"	1000	0.005
Finish Cut	HSS/Carbide	0.025"	900 - 1200	0.002 - 0.005
Cut Off	HSS/Carbide		1000	0.003 - 0.004
Threading	Carbide		800	0.003 - 0.005

NOTES:

1. Tooth form recommendations:

Top Rake Angle:	0° - 5°
Clearance Angle:	5° - 10°
Side Incidence Angle:	30° - 60°
Tip Radius:	0.020" min

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 a coolant during threading is recommended.

MILLING

Operation	Tool Material	Depth of Cut	Speed (sfpm)	Feed (in./rev.)
Face Milling - Rough	HSS/Carbide	0.150"	1100 - 1600	0.015
Face Milling - Finish	HSS/Carbide	0.050"	1400 - 2000	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:	10° - 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 - 300	0.002 - 0.005
Drilling	≥1" dia.	HSS/Carbide	150 - 300	0.008 - 0.020
Reaming		HSS/Carbide	250 - 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° with a relief lip clearance of 5° - 10° and 10° - 20° 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.
5. Coolants (air or liquid) are recommended during drilling and reaming.



NYTEF PLASTICS

800-646-9833

sales@nytef.com