

# Group 6: Glass and Carbon Fiber Reinforced Materials

## GENERAL NOTES FOR MACHINING STOCK SHAPE PRODUCTS THAT ARE REINFORCED WITH GLASS AND CARBON FIBERS:

1. Compressed air or water soluble liquid coolants are recommended when drilling, reaming, tapping, threading, or cutting very thick materials.
2. Glass and carbon fibers are very abrasive to machining tools. High Speed Steel (HSS) tooling will quickly dull if it is used to cut these materials. For short run machining jobs, carbide tools will provide adequate performance but they too will dull quickly. For large production jobs, diamond coated tooling is required.
3. Glass and carbon fiber reinforced materials tend to be extremely brittle and are very susceptible to cracking. Care should be taken during machining to not induce unnecessary stresses. This is especially true during drilling. Always drill pilot holes for any hole with a finished dimension of 1" diameter or larger.
4. Fiber reinforced stock shape materials tend to warp when they are machined into non-symmetrical parts. To minimize warpage, a.) remove material in small steps and b.) utilize 3 day "relaxation" period and/or a secondary annealing cycle after machining the part to 80% of its finished dimensions.
5. For parts that exhibit severe cracking problems, the material can be slowly heated (1 hour per 1/2" of cross section) to 220°F - 240°F prior to machining. At these elevated temperatures, the material will exhibit improved toughness and a lower tendency to crack. Please note that as the material cools, it will shrink do to thermal contraction. To maintain tight tolerances, all finish cuts must be completed at room temperature.

### BAND SAWING

Material Thickness	Pitch (teeth/in.)	Speed (sfpm)
<1"	6 - 14	5000
1" - 3"	3 - 5	4000
3" - 4"	2 - 3	3000

### 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:          | 5° - 15°  | 5° - 15°      |
| Clearance Angle:     | 20° - 30° | 15° - 20°     |
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/Diamond	0.150"	500 - 800	0.005 - 0.015
General Turning	Carbide/Diamond	0.025"	600 - 1200	0.005
Finish Cut	Carbide/Diamond	0.025"	600 - 1200	0.002 - 0.005
Cut Off	Carbide/Diamond		1000	0.003 - 0.004
Threading	Carbide/Diamond		800	0.003 - 0.005

### NOTES:

1. Tooth form recommendations:

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

2. 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	Carbide/Diamond	0.150"	500 - 800	0.020
Face Milling - Finish	Carbide/Diamond	0.050"	600 - 800	0.005
End Milling - Rough	Carbide/Diamond	0.250"	280 - 450	0.002 - 0.010
End Milling - Finish	Carbide/Diamond	0.050"	320 - 520	0.001 - 0.005

### NOTES:

1. Tooth form recommendations:

Rake Angle:	5° - 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.	Carbide	200 - 400	0.005 - 0.012
Drilling	≥1" dia.	Carbide/Diamond	200 - 400	0.008 - 0.020
Reaming		Carbide	250 - 450	0.005 - 0.015

### NOTES:

- 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.
- For small holes (<1" dia.) carbide twist drills with a low helix are recommended.
- For larger holes (≥1" dia.), use a low helix bit with a point angle of 70°-90° with a relief lip clearance of 5°-15° and 0° - 15° rake angle. To minimize cracking, a 1/2" pilot hole should be drilled prior to finish drilling to the required size.
- 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.
- Coolants (air or liquid) are recommended during drilling and reaming.



**NYTEF** PLASTICS

**800-646-9833**

sales@nytef.com