POLYMER INDUSTRIES
GEARED TOWARDS EXCELLENCE

Fabrication Guidelines

GENERAL NOTES:

1. Key properties of plastics that should be kept in mind during the machining of plastic stock shapes are:
   a. Plastics have much higher coefficients of thermal expansion than metals.
   b. Plastics have much lower melting points than metals.
   c. Plastics have lower thermal transfer coefficients than metals.
   d. Plastics are substantially softer than metals.
   e. Some plastics will absorb significant amounts of moisture. This will soften the plastic and cause it to swell.
   f. Common cleaners and coolants will cause some plastics to craze, crack, and/or become brittle.

2. Most plastic materials can be machined with the same tools and methods that are used for machining soft metals like aluminum and brass. Because plastics are much softer than metals, material can be removed from plastic work pieces at rates that are several times the rates typically utilized with metal pieces.

3. Due to the softness and high thermal expansion rates of plastics, typical tolerances for machined plastic parts are 10 times that typically used for metals. For example, typical machining tolerances for acetal parts are a minimum of+/- 0.001" per inch of part dimension.

4. Because plastic is an excellent thermal insulator, heat generated during the machining operation does not dissipate through the work piece. Excessive heat can affect the surface finish of the work piece, prematurely dull the tool, and in the worst case, melt the work piece. Dull tools generate frictional heat. To avoid heat build up, keep cutting tools as sharp as possible.

5. For most machining of non-reinforced plastic materials, High Speed Steel (HSS) cutting tools are sufficient. Carbide tools are recommended for machining of plastic materials reinforced with abrasive fillers (glass fibers, glass beads, and carbon fibers). For large volume production jobs on plastics with abrasive fillers, diamond coated tools may prove to be economical because of the added life that they provide.

6. High Speed Steel (HSS) cutting tools can be ground to a sharper edge than carbide tools, but HSS will not maintain an edge as long as carbide. Carbide tools with ground peripheries and polished surfaces minimize frictional heat generation and promote chip removal.

7. Always use tools with positive geometries and adequate chip clearance to prevent chip build up.

8. For most plastics machining operations, fast tool speed and slow material feed is recommended.

9. Surface Feet Per Minute (SFPM) = 0.262 x diameter (in.) x RPM.

10. During turning, the best surface finishes are produced by using a broad nosed tool with a rounded end.

11. Most plastic materials can be milled and turned without the use of coolants. For a.) all drilling operations and
    b.) machined parts that require premium surfaced finishes and tight tolerances, compressed air or water soluble coolants can be used. Please note: Traditional petroleum based coolants/cutting fluids can degrade some plastics. Always check the chemical compatibility of the plastic material you are machining before putting it in contact with any chemical solution.

12. Plastic materials are significantly softer than metals and high tool pressures will deflect plastic work pieces away from the cutting tools. Always assure that work pieces are adequately supported to prevent excessive deflections during machining.

13. To minimize internal stresses and assure the highest levels of dimensional stability, all plastic stock shapes manufactured by Polymer Industries are annealed during manufacturing. However, due to the morphology created by filler/ reinforcement packages, some stock shape materials (example: glass fiber filled nylon) tend to retain significant levels of residual stresses even after factory annealing. These materials should be pre-machined to the approximate shape and dimensions of the finished part and then set aside and allowed to "relax" for two to three days. The parts can then be finish machined to their final dimensions and tolerances. In worst case scenarios, parts may require a secondary annealing cycle after pre-machining and prior to final machining.

14. Machined plastic parts that are non-symmetrical will always tend to warp towards the side with the least material. This warpage can be minimized by:
    a. removing material from the opposing sides of the part in gradual steps. When a large amount of material is to be removed, remove 1/4" from one side and then flip the part and remove 1/4" from the other side. Repeat the process until the machining if complete.
    b. machining the part to 80% of it's finished dimensions and then re-anneal the semi-completed part before machining to the finished dimensions.
    c. utilizing a material that is less prone to warpage. Unfilled materials warp less than composites and amorphous materials usually warp less than crystalline materials.

15. Some plastics (ex: Nylon) will absorb significant amounts of moisture when they are in contact with liquids and also directly from humid air. This absorbed moisture will cause the plastic material to swell. This swelling should be considered when part tolerances are determined.

POLYMER INDUSTRIES
GEARED TOWARDS EXCELLENCE

1-877-POLYMER (765-9637)

Machining Index

			Relative Machinability (1 = easiest)	Recommended Cutting Tool Material(s)
GROUP 1: ACETAL PRODUCTS				
a.	Acetal H	Delrin® 150E Homopolymer	1	HSS/Carbide
b.	Acetal H BK	Black Delrin 150E Homopolymer	1	HSS/Carbide
c.	Acetal C	Copolymer Acetal	1	HSS/Carbide
d.	Acetal C BK	Black Copolymer Acetal	1	HSS/Carbide
e.	Acetal H Lf13	13% PTFE Powder filled Homopolymer	1	HSS/Carbide
f.	Delrin 100AF-13	13% PTFE Fiber filled Homopolymer	1	HSS/Carbide
g.	Delrin DE-588	U.S. Navy Use Only	1	HSS/Carbide
h.	Acetal esd	Static Dissipative Copolymer Acetal	1	HSS/Carbide
GROUP 2: NON-REINFORCED EXTRUDED NYLON PRODUCTS				
i.	Nylon	Natural Nylon 6/6, Unfilled	2	HSS/Carbide
j.	Nylon BK	Black Nylon 6/6, Unfilled	2	HSS/Carbide
k.	Nylon Ld	MoS2 filled Nylon 6/6	2	HSS/Carbide
l.	Nylon Im	Nylon 6/12	2	HSS/Carbide
m.	Nylon im	Zytel® ST801	2	HSS/Carbide
GROUP 3: MISCELLANEOUS SPECIALTY ENGINEERING MATERIALS				
r.	PBT	Polybutylene Terephthalate	4	HSS/Carbide
s.	PET	Polyester Terephthalate	5	HSS/Carbide
t.	PVDF	Type 740 Fluoropolymer	4	HSS/Carbide
u.	Nylon Rk	Kevlar® Fiber Filled Nylon 6/6	3	HSS/Carbide
GROUP 4: NON-REINFORCED AMORPHOUS HIGH PERFORMANCE MATERIALS				
v.	PC	Polycarbonate	3	HSS/Carbide
w.	PPO	Noryl® Polyphenylene Oxide and Styrene	3	HSS/Carbide
x.	PEI	Ultem® Polyetherimide	6	HSS/Carbide
y.	PSU	Polysulfone	5	HSS/Carbide

Machining Index

			Relative Machinability (1 = easiest)	Recommended Cutting Tool Material(s)
GROUP 5: NON-REINFORCED CRYSTALLINE HIGH PERFORMANCE MATERIALS				
z.	PEEK	Polyetheretherketone	4	Carbide
GROUP 6: GLASS AND CARBON FIBER REINFORCED MATERIALS				
aa.	Nylon Rg	Glass Fiber Filled Nylon 6/6Glass	7	Carbide/Diamond
bb.	PC Rg	Fiber Filled Polycarbonate	7	Carbide/Diamond
cc.	PPO Rg	Glass Fiber Filled Noryl	6	Carbide/Diamond
dd.	PEI Rg	Glass Fiber Filled Ultem	8	Carbide/Diamond
ee.	PEEK Rg	Glass Fiber Filled PEEK	7	Carbide/Diamond
ff.	PEEK Rc	Carbon Fiber Filled PEEK	7	Carbide/Diamond
gg.	PEEK L	Bearing and Wear Grade PEEK	7	Carbide/Diamond

Group 1: Acetal Products

GENERAL NOTES FOR MACHINING ACETAL STOCK SHAPE PRODUCTS:

1. Compressed air or water soluble liquid coolants are recommended when drilling, reaming, tapping, threading, or cutting very thick materials.
2. Although Acetal is one of the most forgiving thermoplastics materials to machine, achieving very high quality surfaces may require experimentation with tools, feeds and speeds.
3. Acetal is made by polymerizing formaldehyde. The smell of formaldhyde during machining indicates excessive heat generation. Review feeds, speeds, and the sharpness of cutters to minimize the heat being generated.

BAND SAWING

Material Thickness	Pitch (teeth/in.)	Speed (sfpm)
<1"	4 - 6	300 - 400
1" - 3"	3 - 4	300 - 400
3" - 4"	2 - 3	300 - 400

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: 0°- 8° 0°-10°
   Clearance Angle: 30°- 40° 10°-15°
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"	450 - 600	0.010 -0.020
General Turning	Carbide	0.100"	600	0.010
Finish Cut	HSS/Carbide	0.025"	600 - 800	0.003 - 0.007
Cut Off	HSS/Carbide		800	0.003 - 0.004
Threading	Carbide		500	0.005 - 0.010

NOTES:

1. Tooth form recommendations:
   Top Rake Angle: 0°- 5°
   Clearance Angle: 15° - 20°
   Side Incidence Angle: 15° - 20°
   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"	1400 - 2000	0.020
Face Milling - Finish	HSS/Carbide	0.050"	2500 - 3000	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°-10°
   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	200 - 500	0.004 - 0.015
Drilling	≥1" dia.	HSS/Carbide	150 - 300	0.010 - 0.050
Reaming		HSS/Carbide	350 - 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 3°-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.
5. Coolants (air or liquid) are recommended during drilling and reaming.

Group 2: Nylon Products

GENERAL NOTES FOR MACHINING NON-REINFORCED EXTENDED NYLON PRODUCTS:

1. Compressed air or water soluble liquid coolants are recommended when drilling, reaming, tapping, threading, or cutting very thick materials.
2. 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 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°-5°
   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"	HSS/Carbide	150 - 250	0.010 - 0.050
Reaming	dia.	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.

Group 3: Miscellaneous Specialty Engineering Materials

GENERAL NOTES FOR MACHINING MISCELLANEOUS SPECIALTY ENGINEERING MATERIALS:

1. Compressed air or water soluble liquid coolants are recommended when drilling, reaming, tapping, threading, or cutting very thick materials.
2. Group 3 Materials 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 Group 3 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.
4. While not required, the use of chilled coolants will improve chip removal and allow higher feed rates.

BAND SAWING

Material Thickness	Pitch (teeth/in.)	Speed (sfpm)
<1"	6 - 8	4000
1" - 3"	4 - 5	3000
3" - 4"	2 - 3	2000

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:	0°-8°	0°-10°
Clearance Angle:	30°-40°	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	0.150"	450 - 600	0.005 - 0.015
General Turning	Carbide	0.100"	600	0.005
Finish Cut	HSS/Carbide	0.025	600 - 800	0.002 - 0.005
Cut Off	HSS/Carbide		800	0.003 - 0.004
Threading	Carbide		500	0.003 - 0.005

NOTES:
1. Tooth form recommendations:
   Top Rake Angle: 0°-5°
   Clearance Angle: 5°-10°
   Side Incidence Angle: 30°-60°
2. Single or dual fluted cutters are desirable because they produce less heat and vibration than multi-fluted helical cutters.

MILLING

Operation	Tool Material	Depth of Cut	Speed (sfpm)	Feed (in./rev.)
Face Milling - Rough	HSS/Carbide	0.150"	1100 - 1600	0.020
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: 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 - 300	0.002 - 0.005
Drilling	≥1"	HSS/Carbide	150 - 250	0.008 - 0.020
Reaming	dia.	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°-130° with a relief lip clearance of 5°-10° 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.
5. Coolants (air or liquid) are recommended during drilling and reaming.

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"	HSS/Carbide	150 - 300	0.008 - 0.020
Reaming	dia.	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.

Group 5: Non-Reinforced Crystalline High Performance Materials

GENERAL NOTES FOR MACHINING NON-REINFORCED CRYSTALLINE STOCK SHAPE MATERIALS:

1. Compressed air or water soluble liquid coolants are recommended when drilling, reaming, tapping, threading, or cutting very thick materials.
2. Some crystalline stock shape plastics are somewhat abrasive. While low volume machining can be efficiently completed with High Speed Steel (HSS) cutters and tooling, re-sharpening can be significantly reduced by using carbide tooling on large volume production jobs.
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	4500
1" - 3"	4 - 5	3500
3" - 4"	2 - 3	2500

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°	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: 5°-10°
   - Clearance Angle: 0°-10°
   - Side Incidence Angle: 40°-70°
   - 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	Carbide	0.150"	500 - 800	0.015
Face Milling - Finish	Carbide	0.050"	600 - 800	0.005
End Milling - Rough	HSS/Carbide	0.250"	300 - 500	0.002 - 0.010
End Milling - Finish	HSS/Carbide	0.050"	400 - 600	0.001 - 0.005

NOTES:

1. Tooth form recommendations:
   - Rake Angle: 5°- 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	200 - 400	0.002 - 0.005
Drilling	≥1" dia.	HSS/Carbide	200 - 400	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 118° 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.

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: Side 5°-10°
   - Incidence Angle: Tip 5°-10°
   - 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:

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. For small holes (<1" dia.) carbide twist drills with a low helix are recommended.
2. 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.
3. 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.
4. Coolants (air or liquid) are recommended during drilling and reaming.

POLYMER INDUSTRIES
GEARED TOWARDS EXCELLENCE

1-877-POLYMER (765-9637)

sales@polymerindustries.com