



DRAKE
PLASTICS TM, Co.

Machining Guidelines – Torlon

Machining Tidbits...

- Torlon shapes are stronger and stiffer than most plastics but considerably softer than most metals meaning fixturing is critical.
- All Torlon grades are abrasive on tooling. Carbide tooling can be used for short runs. Polycrystalline (PCD) tooling should always be considered long runs, tightly tolerance parts and for reinforced grades.
- Torlon has lower elongation than many other plastics. Excessive tool rubbing during machining can cause material pullout that resembles localized porosity. Deep hole drilling without sufficient coolant can lead to cracking.
- All extruded Torlon shapes have a cured outer skin. This layer is .010-.020" thick. It is harder than interior sections and can be machined off or left on. It does offer the best wear and chemical resistance.
- Machined Torlon parts can be recurred after machining if maximum wear resistance and chemical resistance is required. Contact Drake prior to ordering if a post cure is planned.

Turning

Positive geometries with ground peripheries are suggested for inserts. Fine grained C-2 carbide or PCD inserts are best.

360° chuck pressure is suggested to avoid distortion. Pie jaws and or soft jaws should be used when turning thin walled tubular shapes.

Drilling

Low helix drill bits and flood coolant are best for holes below 1" diameter. Peck drilling is suggested for swarf removal.

Larger diameter holes are best approached using a 2 step process incorporating a drilled pilot hole (1/2" diameter max) and a boring to finish diameter.

Threading

Single point inserts with flood coolant should be used for threading. Two fluted, coated taps and suggested for tapped holes. Tapping should be done with coolant.

Milling

Part fixturing is critical for milling as high spindle speeds and fast travel are preferred to minimize frictional heat buildup and material pullout. Cutters should be designed with positive geometry. Climb milling is recommended over conventional milling.

Torlon[®] Machining Parameters

Turning

| | | |
|----------------------|-----|-----|
| Cutting speed, m/min | 90 | 240 |
| Feed, mm/rev | 0,1 | 0,6 |
| Relief Angle, ° | 5 | 15 |
| Rake Angle, ° | 7 | 15 |
| Cutting depth, mm | 0,6 | |

Circular sawing

| | | |
|----------------------|---------------|------|
| Cutting speed, m/min | 1830 | 2440 |
| Feed, mm/rev | fast & steady | |
| Relief Angle, ° | 15 | |
| Set | slight | |
| Rake Angle, ° | 15 | |

Milling

| | | |
|----------------------|------|-----|
| Cutting speed, m/min | 150 | 240 |
| Feed, mm/rev | 0,15 | 0,9 |
| Relief Angle, ° | 5 | 15 |
| Rake Angle, ° | 7 | 15 |
| Cutting depth, mm | 0,9 | |

Drilling

| | | |
|----------------------|-----|-----|
| Cutting speed, m/min | 90 | 240 |
| Feed, mm/rev | 0,1 | 0,4 |
| Relief Angle, ° | 0 | |
| Point Angle, ° | 118 | |

Reaming

| | | |
|-----------------|-----|--|
| Slow speed, rpm | 150 | |
|-----------------|-----|--|

NOTE: PCD Coated Carbide or sharp Carbide tools are recommended for optimal results.