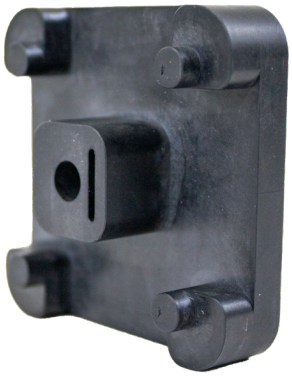
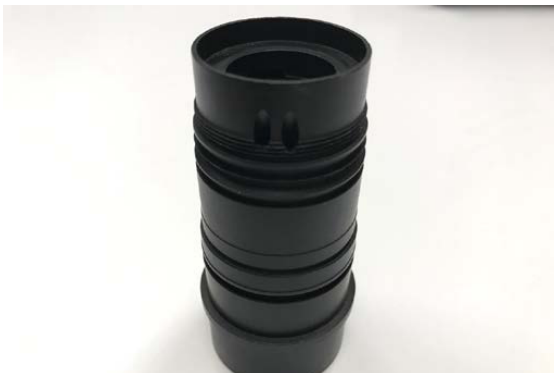
MACHINING GUIDE – RYTON® PPS

MACHINING NOTES – RYTON:



- **Fixturing is critical:** Ryton shapes are stronger & stiffer than most plastics, but considerably softer than most metals.
- **All grades are more abrasive on tooling than softer plastics like nylon and acetal:** This is especially true of the R4 grade, which is reinforced with 40% glass fiber.
    - **Short runs** – consider carbide tooling
    - **Long runs/Tight tolerances/Reinforced grades** – always consider Polycrystalline (PCD) tooling
- **Lower elongation than many other plastics:** Deep hole drilling into heavy cross sections without enough coolant may lead to cracking.



- **Coolant can be used while machining Ryton:**
    - Both water soluble & petroleum-based coolants may be used.
    - Appropriate use of coolants will extend tool life and improve surface finish.
    - Air (preferably from a cold air gun) can also be used for small parts including those in which clean-up is difficult.

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. Machined soft jaws or pie jaws should be used when turning thin-walled, tubular shapes. Rough turning the chuck area of the stock is suggested to improve roundness.
- Internal plugs should be used to prevent thin-walled parts from compressing and distorting.

DRILLING

- Care to minimize heat build-up is important when drilling holes that are more than 2X the diameter.
- Low helix drill bits and flood coolant are best for drilling holes. ***Peck drilling*** is suggested for swarf removal. Coolant fed drills are ideal for removing swarf and preventing excessive heat build-up.
- Larger diameter holes are best approached using a 2-step process incorporating a drilled pilot hole (½" diameter maximum) and boring to finish diameter. Drilled holes ½" diameter and smaller can be machined using a standard carbide drill. Holes up to 2" diameter can be machined using an insert drill, such as Iscar Chamdrill.
- Pocketing is suggested for mill set ups. To avoid breakout of the back side, consider milling from both sides or leaving .005-.010" that you remove by milling with a small end mill.

THREADING

- Single point inserts with flood coolant should be used for threading during turning. Two fluted, non-coated spiral carbide taps are suggested for tapped holes.
- Tapping should be done with a cutting fluid. Tight tolerance tapped holes may require 1-size larger tap than usually needed to tap aluminum or steel.
- When thread milling, floating tap heads can minimize tap breakage common with smaller sizes taps.

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***, as it provides better chip removal, lower tool wear, and better surface finish.
- End mills with 4 flutes should be used when possible.
- During milling, step overs should be limited to 25% tool diameter and depths of cut 50% of tool diameter to achieve an optimal surface finish.

SAWING

BAND SAWING **is the preferred method of cutting Ryton shapes.** It can be used for both straight and contoured cuts of plate, in addition to rod and tubular bar.

- *Saw blades should be chosen based on material thickness and precision. They also must have enough clearance to minimize heat build-up. Triple chip blades 2.5-3.5 teeth per inch are suggested. We also have good results with .035" thick x 1" wide blades.*
- *Using fewer teeth per inch than metals typically require will help reduce heat build-up. We suggest 3 teeth per inch at a band saw speed of 2500 ft per minute as a starting point.*
- *Coolant (fluid and/or air) should be used.*

TABLE SAWING can be used, but care must be exercised to ensure safety. Residual stress within shapes can cause material to close in on the blade. When using a table saw, partial cuts into the thickness are best.

- *Rip and combination blades with carbide tips are suggested. We suggest fewer teeth per inch than would be used on metals or wood.*
- *A 60 teeth 12" diameter rip and combination blade should yield smooth cuts on plates up to 1/2" thickness.*

CHOP SAWS and radial arm saws may be used, but care must be exercised to ensure safety. Residual stress within shapes can cause material to close in on the blade. When using a chop saw, repeated partial cuts are required to minimize heat buildup when cutting cross sections greater than 2".

- *Rip and combination blades with carbide tips are suggested. We suggest fewer teeth per inch than would be used on metals or wood.*
- *A 60 teeth 12" diameter rip and combination blade should yield smooth cuts.*

Ryton® Machining Parameters

TURNING		
Depth of Cut	Speed	Feed
0.025"	300 - 800 ft./min	0.004 - 0.025 in/rev

DRILLING	
Hole Diameter	Feed
0.0625"	0.007 in/rev
0.125"	0.010 in/rev
0.250"	0.012 in/rev
0.500"	0.015 in/rev
0.750" & up	0.020 in/rev

FACE MILLING		
Depth of Cut	Speed	Feed
0.035"	500 - 800 in./min	0.006 - 0.035 in./tooth

END MILLING			
Tool Size	Depth of Cut	Speed	Feed
1/4	0.250"	270 - 450 ft./min	0.002 in./tooth
1/2	0.250"	270 - 450 ft./min	0.003 in./tooth
3/4	0.250"	270 - 450 ft./min	0.005 in./tooth
1.0	0.250"	270 - 450 ft./min	0.008 in./tooth

SAWING	
Circular Saw	Bi-Metal Blade
Carbide Blade, 3000 - 4000 RPM	Bi-Metal Blade, 3000 - 4000 ft./min