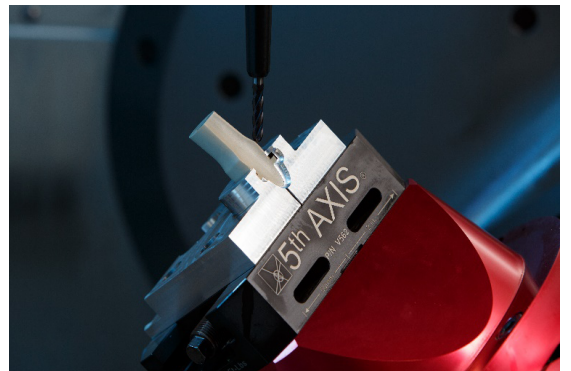




MACHINING GUIDE – PEEK

MACHINING NOTES – PEEK:

- **Fixturing is critical:** PEEK 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 or acetal:** Especially true for grades reinforced with glass and/or carbon 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 cause cracking.
- **Coolant can be used while machining PEEK:**
 - 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 where 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 PEEK 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.*

PEEK® 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 |