TriStar

Engineered Plastic Solutions™

Machining Plastics:

The Essential Guide to Materials, Tools and Techniques

machining plastics

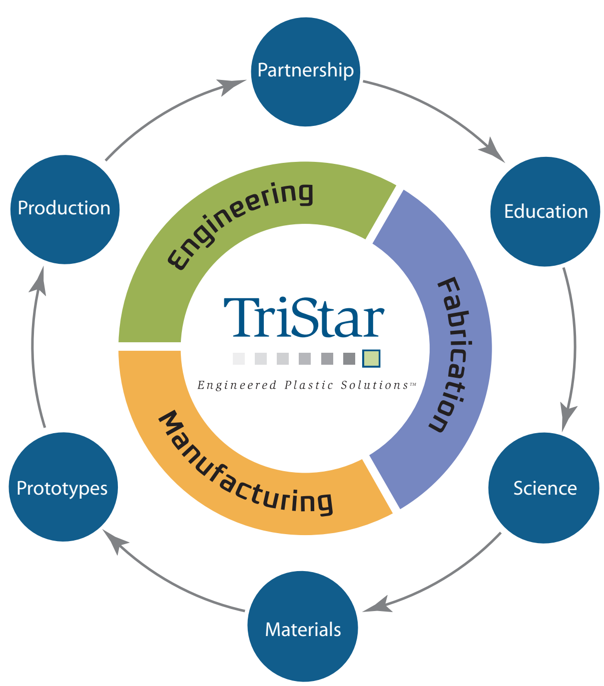
Table of Contents

TriStar Delivers Plastics Machining Expertise

TriStar Plastics offers complete plastic manufacturing, machining, surface modification and distribution — we are your source for one-stop-shopping of engineered plastics. With the latest CNC machining, turning and milling equipment, we can guarantee your parts will meet design specs and are fully inspected and certified. We can also help you save on component costs by suggesting alternate materials, or providing machining tips to help you reduce scrap; we've built a solid reputation in over 70 industries. And since we do it all in-house, we'll help you reduce fabrication delivery time so that you can meet your production deadlines.

In addition to the one-on-one support we offer, we have a number of online resources at tristar.com you can take advantage of. They include:

- Our interactive Material Database
- Design Worksheets
- Instructional video library
- Technical email updates
- Monthly technical briefs
- Direct support from TriStar engineers via our Ask the Expert form.



TriStar
Engineered Plastic Solutions™

TriStar offers the latest machining equipment staffed by experienced operators trained in the latest techniques to help you make the most of your material investment.

Plastic is not metal.

This is the first lesson many fabricators discover when attempting to machine plastics for the first time. While both materials are technically "machinable," the similarities end there.

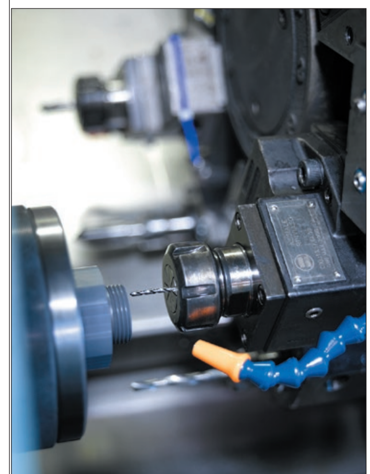
Metals are generally pure materials, while plastics are a hybrid of different components. Whereas metals retain their shape and have a predictable melting point, plastics can expand to five (or more) times their original dimension and offer varying heat tolerances. Machining metals follows a predictable pattern with minimal creep. When machining plastics, quick adjustments must be made to accommodate substantial creep — not to mention that the material has a strong propensity for chipping and melting during machining.

Simply stated, the basic principles of machining metals do not apply when machining plastics.

Machining Plastic has Unique Challenges ... and Rewards

With the right material selection, proven handling techniques, plus the proper tools and coolants, machining plastic parts is not only attainable, but achievable by many machine shops.

The goal of this technical guide is to demystify the art of machining plastics. We'll explore plastic properties, selection criteria, price points, expansion rates, tolerances, and nuances of material and tool selection and review machining techniques. Because when you fully understand the significant differences between machining plastics vs. machining metals, you can improve your design and, ultimately, the quality and performance of your product.



Plastic materials are challenging to machine given substantial creep, varying heat tolerances and a propensity for chipping and melting.

Material Selection: Cost vs. performance

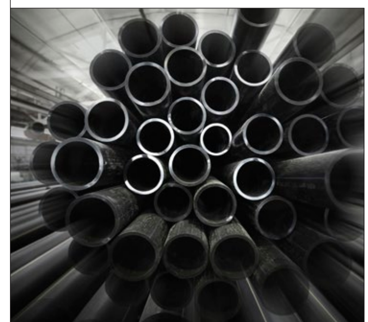
How do you select the ideal material for your application? There's still a widespread belief that "traditional" metals outperform plastics, when actually the opposite is true. Plastics are an excellent replacement for bronze, stainless steel, and cast iron, and they excel in high-temperature and extreme working environments.

But this high level of performance comes at a cost. Plastics are not "the cheap stuff," and some high-performance formulas are substantially more expensive than metal. For example, Polybenzimidazole (PBI-Celazole) is 25x the price of cold-rolled steel, and 15x more costly than Type 303 stainless steel. Given these price points, it is critical to employ expert machining techniques to use costly materials efficiently and reduce scrap.

Ultimately, the decision of material type should come down to *an investment in performance*. Choosing a higher-quality material will yield a higher-quality part. And higher-quality parts can save you from in-field failures or costly recalls down the line. Better to invest up-front and avoid these hazards.

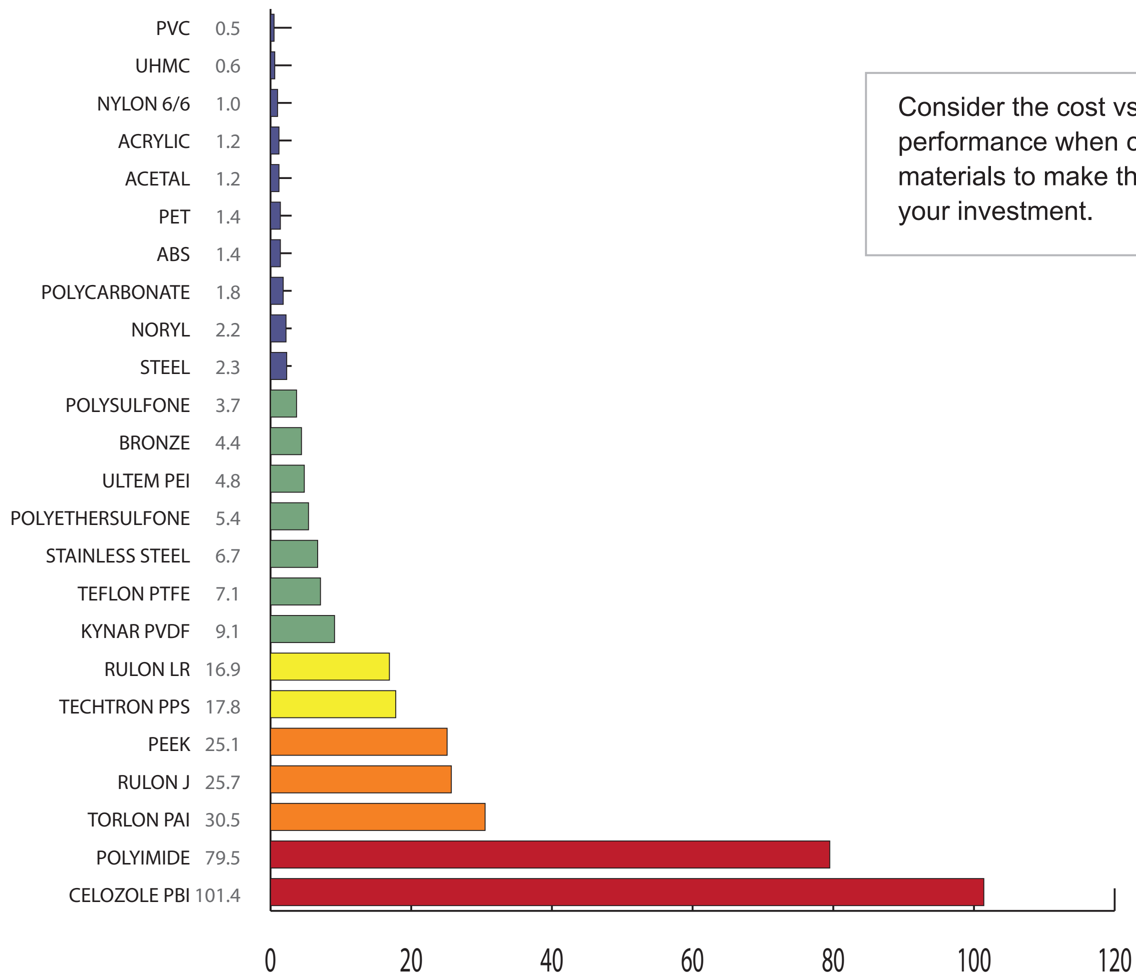
When should you choose plastic over metal materials? Consider the advantages of plastic machined parts, they have the ability to:

- Reduce component weight
- Eliminate corrosion
- Lower noise level
- Improve wear performance
- Extend service life
- Insulate and isolate (thermally and electrically)



Plastic materials are a superior replacement for traditional metals, bronze, stainless steel and cast iron.

Relative Cost of Plastic Materials



Consider the cost vs. performance when choosing materials to make the most of your investment.

Material Selection: Thermoset vs. Thermoplastic

Now that we've established the costs associated with plastic materials, the question then becomes which category of plastics should you choose?

Thermoset plastics retain their solid state indefinitely and include just a few trade names. Thermoplastics can be melted more than once to form new shapes and comprise the largest group of plastics. They are also the type best suited to machining. Don't be fooled by similar-sounding names; as each "thermo" category boasts unique characteristics.

Thermoset plastics:

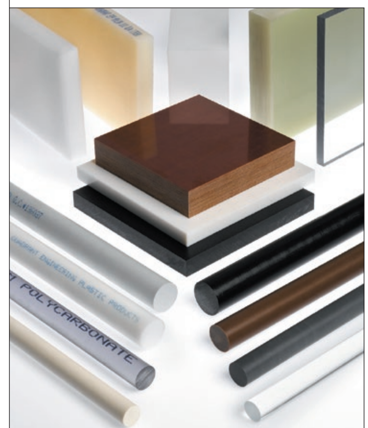
- Do not melt since they chemically change in molding
- Are usually brittle and chip easily
- Often incorporate fillers as part of a composite
- Common formulas:
  - Phenolic
  - Epoxy
  - PTFE
  - Micartas
  - Melamines

Thermoplastics:

- Largest class of plastics
- Melt and reform without changing chemically
- Include a diverse list of trade and generic names including:
  - Acetal, Acetal, ABS, Nylon, Polyethelene, PVC, Teflon
- Filler options include:
  - Glass fibers, Carbon fibers, Graphite, Carbon, Molybdenum disulfide, PTFE

In an industry where brand name recognition can lead to an automatic material order, beware of the plastic material "name game" — where each processor names "their" material for what is essentially a trade product. For instance, the material Acetal is a generic material, yet there are several different market names for it. DuPont calls its version Delrin®. Hoechst uses the name Hostaform®. Celcon® is the Celeanse trade name, and Quadrant calls certain Acetal versions Acetron®, while Ensinger-Hyde uses the name Hydex®. That marks five different names for a single product — no wonder there is confusion in the marketplace!

To learn more about the hazards of unknowingly purchasing counterfeit materials, check out our free companion paper, Rulon Bearings: How to Recognize Genuine and Avoid Counterfeit.



While cost is important, ultimately, materials should be selected based on their application performance and true trade name.

Why machine plastics?

Once you've selected the proper plastic material, the next question becomes one of machining vs. injection molding. Most plastic components are produced via injection molding, which is the most cost-effective method. But machining is the better fit based on:

- **Low initial costs** – molding equipment requires a large initial investment in tooling equipment. Machining is more economical for lower volumes and prototypes.
- **Tight tolerances** – machining allows for much tighter dimensional tolerances than can be achieved with injection molding.
- **Physical property limitations** – some materials such as PTFE and UHMW are impossible to mold and require machining.
- **Stress factors** – injection molded parts are subjected to more stress than extruded rod, tube, and sheet. Machining will produce more consistent results.

Typical applications for machining plastics include semiconductor processing components, heavy equipment wear parts, and food processing components.

Machining Plastics 101: Limit Heat!

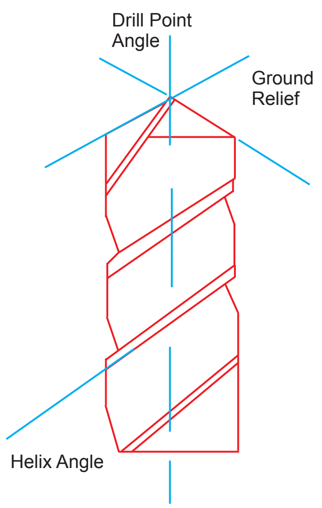
The most important consideration in machining is to limit the amount of heat buildup, as the very act of machining generates friction, and thus, heat. Be aware that anytime you machine plastics, your cutting tool can instead become a "melting tool." Heat also presents dimensional challenges, so you must be aware that as a part expands it becomes more difficult to hold tolerances.

With plastics, always consider heat tolerances and keep in mind that drilling generates more heat than any other machining technique.

Drilling Operations

Heat-related changes are more prevalent with some plastics than others. Many of the plastics with high-expansion rates have low-melt temperatures. For instance, UHMW, has an expansion rate 20x that of steel and a melt temperature of 266° F. Fillers add another new level to expansion rates. Unfilled PEEK expands 26 X 10'6 in/in/OF while 30% carbon fiber filled PEEK is 10. In contrast, adding PTFE only to PEEK raises the expansion rate to about 30, yet none of these fillers change the melt temperature.

Drilling



MORE HEAT IS GENERATED IN DRILLING THAN IN ANY OTHER OPERATION

Twist drills with twist angles between 12 and 18 degrees
- Large flute area assists in chip removal

Cutting lip should be ground so one edge is .005" to .010" longer than the other

Use blunt angles (115 to 130 degrees) for thinly walled pieces to prevent expanding the OD

How do I reduce heat while machining plastics?

Intolerance to heat can appear as surface finishes that go from smooth to very rough. Or you may notice small balls of melted plastic on the surface of your component. To reduce the impact of friction-induced heat, consider the following potential causes:

- Cutting speeds & feed rates
- Tool designs
- Cutting tool materials
- Use of coolants
- Sharpness of cutting tools

Drilling tips to maintain heat levels:

- Drills with twist angles of 12°-18° and with large flute areas will help remove chips and heat from the drilling hole.
- Grinding relief onto the drill will also reduce friction. Angles will vary by material, but 20°-50° is a good starting point.
- For softer materials, high-speed steel drills are adequate, but highly-abrasive plastics (filled materials), require carbide (Titanium Nitrite/TiN/AlTiN), CVD (chemical vapor deposition) diamond, or PCD (polycrystaline diamond) tooling.
- Remove the drill from the hole (pecking) frequently to remove chips and give the material a chance to cool slightly.
- Slower RPMs than technically called for can be beneficial depending on the material and other conditions.
- Never use any tool that has already drilled metal, as it is too dull and will impact tolerances and surface finishes.

Drill Size Vs RPM

Drill Size	RPM
No. 60 thru 33	5,000
No. 32 thru 17	3,000
No. 16 thru 01	2,500
1/16	5,000
1/8	3,000
3/16	2,500
1/4	1,700
5/16	1,700
3/8	1,300
7/16	1,000
1/2	1,000
A thru D	2,500
E thru M	1,700
N thru Z	1,300

Turning operations and heat levels

The number one challenge in turning — just as in general machining — is maintaining proper heat levels. Turning operations require inserts with positive geometries and ground peripheries. Ground peripheries and polished-top surfaces generally reduce material build-up on the insert, which can improve the final surface finish. A finely-grained C-2 carbide or PCD is generally the best option for turning operations. Try to mill the slot across the outer diameter to break up chips.

Plunge cutting or peck (interrupted cut) drilling is a good way to remove material and to provide dimensional repeatability, but there are a few rules to follow.

Turning tips for form or plunge cutting:

- Insert the tool width at less than the minimum diameter of the component
- Consider feed rates, which are dependent on the stiffness of the stock (but generally 0.004 TO 0.010"/REV)
- Surface finish at the bottom of the cut is best controlled by approaching the bottom of the cut slowly, reaching the bottom, and clearing the tool immediately. Use the smallest width possible to turn across.
- Do not dwell at termination, or you may experience drag that alters the surface finish.
- When turning (lathing), use single-point or partial threading inserts. This results in cleaner threads and provides ample room for chips.
- For milling, use single-form thread milling cutters for soft materials; multi-form for harder materials.

Threading and Tapping

All plastics are notch sensitive, meaning that small sharp "V" threads can cause problems such as tearing. By putting a chamfers on the rod ends or into the holes before a threading operation, you can reduce the tendency of the initial thread to tear. We often recommend the use of coolants when threading and tapping. And remember that any instrument that has tapped metal is not sharp enough to tap plastics.

Threading and tapping tips:

- Threading is best achieved with a single point using a carbide insert and taking four to five 0.001" passes at the end.
- Use H-3 for smaller diameters, H-5 for larger.
- There are +.003"/.005" oversized taps available that can achieve a qualified thread size with softer materials. Many soft materials will expand out, then close back in when tapped. Thread-milling gives you better size control when the size and depth is friendly for the feature.
- Two flute taps with enlarged flutes will help remove chips and keep the taps clear.
- If the tapped area must withstand heavy stresses or continued insertion and removal of connectors, the use of metal threaded inserts is preferred over a tapped plastic piece.
- Inserts can be pressed into place, ultrasonically inserted, or threaded into the plastic using self-tapping features.
- Ultimately, the structural integrity of the material (hard or brittle) will determine the best insert type.

Milling and Cutting

When it comes to milling plastics, climb milling is recommended over conventional milling. And the most difficult challenge is in keeping the component from moving or vibrating during operation, which can result in chatter marks on the components. To maintain control, we often employ vacuum systems (which require a flat surface) or fixture clamps (which seem to always get in the way). But be aware that these methods are acceptable as long as they do not stress or distort the piece. For best results, we often recommend double-sided adhesive tape to prevent parts from moving.

Other work-holding methods include building holding tools from excess material, making drill-through holes for top clamps and nuts, board mounts, and vacuum chucks (these are often built into CNC routers).

Tips for milling with adhesive tapes:

- Completely clean both the machine surface and the work component before beginning.
- Make sure the surface of the work piece is completely covered in tape.
- Place the piece onto the machine surface as quickly as possible after removing the protective layer.
- Tap the piece with a rubber mallet to insure it is securely seated
- To remove the completed piece it may be necessary to dissolve the adhesive with alcohol and pry apart carefully.

Beware of burrs

A common hazard of milling is burrs, which are created when a tool reaches a travel end and the plastic piece is not supported. To eliminate burrs, you can bring in a second material to the edge of the work piece so that the cutting tool continues into this secondary material (which also reduces chipping). This will allow a clean cut right to the edge.

Increasing the amount of chamfering used on the piece (within reason) lets the machine do much of the work.

To remove burrs, consider:

- Tumbling parts against each other
- Tumbling parts in media
- Sanding
- Polishing wheels
- Removing burrs by hand with specialized tools

The same concept for burring also applies to milled surfaces in general. If you must mill a slot across a cylindrical part, it may save you money to cut the slot in two inside-out cuts rather than one straight-across cut. The time saved in deburring may pay for the longer machining cycle.

Ultimately, the best solution to remove burrs is to *avoid them in the first place,* since you can reduce secondary finishing time and associated costs.

Sawing Operations

Sawing is employed in many machining applications. Band sawing is ideal for straight, continuous and irregular cuts. Table sawing is also convenient for straight cuts of multiple thicknesses or thicker cross sections. Saw blades should be selected based on material thickness and desired surface finish. Choose carbide tipped blades for the best results.

Sawing tips:

- For general sawing, plastic-rated rip and combination blades with a 0° tooth rake and 3°-10° tooth set are best to reduce frictional heat.
- Hollow ground circular saw blades without set will yield smooth cuts up to 3/4” thickness.
- Tungsten carbide blades wear well and provide optimal surface finishes. PCD blades also work very well.

The Coolant Connection

To maintain heat temperatures, coolants are often recommended and employed during machining. However, we've found that in many instances, it is best to avoid water-based coolants in order to achieve a premium surface finish. Petroleum-based fluids are another commonly-used coolant, yet they often contribute to stress fractures in amorphous plastics. Materials such as polyimide and nylon can absorb up to 8% moisture, which can cause extreme swelling of parts.

For the closest tolerance and optimal finish, our machining team is moving away from liquid coolants when possible. Instead, we are employing air-line air blowing, cold air guns, and vacuum suction to assist with chip removal and control the finish. Vacuum offers the advantage of keeping tools cool, plus helping to maintain a dust and odor-free machining environment. Vacuuming is also an essential tool for chip evacuation.

Machining Materials: Case Studies

Now that we've covered the nuances of different machining techniques, we'd like to demonstrate their benefits in real-world applications. Read on to explore how TriStar has solved machining challenges by delivering custom-machined parts in a variety of plastic materials.

Machining UHMW

Ultra-high molecular weight polyethylene (UMHW) is one of the most-commonly machined materials in the plastics family. It is known for supplying superior wear resistance and service life in both wet and dry environments. As with all polyethylene materials, UHMW has a low melting point (270°) and a high coefficient of thermal expansion (120 x10" IN/IN/ÜF).

UMHW is the most commonly machined member of the polyethylene family. The basic material is relatively soft and cuts readily but because the material melts easily, it is especially important that we limit heat build-up. Sharp tools, the application of chips in the work area, and proper tool shape design are particularly important when machining UMHW.

Sawing UHMW	
Cutting Speed	3,000 to 13,000 ft./min
Feed	0.0009 to 0.0040 in/tooth
Rake Angle	0 to 5° HS, 3 to 8° HSS
Clearance Angle	10 to 15° HS, 30 to 40° HSS
Pitch Setting	0.020 to 0.040 in 0.030 to 0.040 in

Common applications of UHMW:

- Food processing and packaging bearings
- Medical materials
- Coal and quarry bushings
- Hot-oil drills

The TriStar Advantage for Machining UHMW: Burr elimination for smooth finish

Challenge:

Our partner is a major producer of microwave synthesis equipment used in medical applications. They were experiencing a problem with machining their UHMW manifold components, which consisted of many intersecting holes. As they machined holes into the material, they caused a large accumulation of burrs, which impacted the surface finish and performance of the final part.

Solution:

TriStar examined the manifold components and how they were machined. They noted that each time a new hole was drilled, burrs would immediately rise to the surface. To reduce this hazard, we recommended adding a sacrificial rod as a back support when drilling the cross holes. This technique allowed for a cleaner component profile and better product performance. Since adding the support rods, our client has completely avoided burrs and also improved the flow of air and liquids through the manifold holes.

Our client also notes they are able to machine components faster, which has resulted in better overall production rates.

Machining Nylon

Nylon is considered an affordable, long-lasting, high-strength alternative to metal that is also easy to machine. Nylon materials can be extruded or cast (filled or unfilled) and easily resist chemicals and corrosion. Common forms are Nylon 66 (extruded) and Nylon 6 (usually cast into large blanks to be machined into parts). Machining nylon requires carbide tooling, and since nylon is the most hygroscopic of the plastics, care must be taken when using coolants. Part swelling and subsequent drying can cause dimensional problems.

When nylon is used in a wet application, it is important to match the same wet environment during machining to hold sizes.

Nylons are available lubricated or unlubricated. General dimensions are limited to 6” rod and 3” plate, cast into rods to 38”, discs to 80”, sheet to 4” thick. Common fillers include glass or carbon fibers.

Common applications of nylon:

- Bushings, bearings and nozzles
- Pistons and valves
- Manifolds
- Food contact parts
- Electrical and pump components
- Wear pads and strips

The TriStar Advantage for Machining Nylon: Reduced scrap and lower labor costs

Challenge:

Our partner manufacturers vacuum hose and handle components which are designed of glass-reinforced and unfilled nylon. The hose/handle combination consists of 20 parts, which became time-consuming and costly to assemble on the manufacturing floor. Our client wanted to reduce the chance of assembly error and save on labor costs by exploring alternative machining techniques.

Solution:

Working with the manufacturer, we integrated components such as the contact and slip rings into a single part. This approach significantly reduced the chance of errors on the assembly line, and has contributed to a better aesthetic of the finished part.

More importantly, this machining technique reduced scrap and lowered production costs.

Machining Acrylic

Acrylic, also known as PMMA or the trade name Plexiglass, provides outstanding optical properties and excellent resistance to abrasion and scratching. The material has a high tensile strength and can easily deflect high temperatures.

Acrylic is an excellent material to form and bend, and is a good candidate for melting and remelting to improve properties. But special attention must be paid to machining temperatures, as the excess heat generated from machining can cause methyl methacrylate (NIMA) to be released. Strong ventilation is required when working with acrylic.

Drilling acrylic requires a drill with a tip ground to 60°-90° included angle, and backing of the material with another work piece to prevent chipping as the drill breaks through.

When sawing, a carbide-tipped blade with a triple-chip grind is best. Teeth should have a clearance of 100°-150° and a rake angle of l0°-50°. With the right blade angle, material will scrape away rather than chipping. Recommended coolant is water to produce a smooth wall geometry.

Turning - Acrylic

Depth of Cut	Speed (ft/min)	Feed (in/rev)
0.150	450/500	.005/.010
0.025	500/600	.004/.007

Face Milling - Acrylic

Depth of Cut	Speed (ft/min)	Feed (in/tooth)
0.150	1300/1500	0.020
0.025	500/600	.004/.007

Drilling - Acrylic

Hole Diameter	Feed (in/rev)
1/16	.002/.005
1/8	.003/.010
1/4	.005/.012
1/2	.008/.015
3/4	.015/.025
1	.020/.050
1 1/2	.020/.050
>2	.020/.050

Common applications of acrylic:

- Optical lenses
- Display cases
- Lighting fixtures
- Liquid manifolds
- Consumer goods

The TriStar Advantage for Machining Acrylic: Plasma pretreatment

Challenge:

Manufacturers of specialized retail cases often require screen printing of their display cases to promote their goods in a retail setting. Acrylic (or PMMA) is a pliable material that can easily bond to itself, but to increase the bond strength of inks or other coatings, manufacturers look to TriStar to increase adhesion properties as part of the machining process.

Solution:

TriStar incorporated plasma surface modification to increase acrylic bond strength. The acrylic/PMMA is subjected to plasma gas mixture to induce an adherent surface to a structural epoxy. Results of treated vs. untreated acrylic bonding strength appear below:

PMMA - Untreated vs Plasma Treated

	Untreated PMMA	Plasma Treated PMMA Process 1	Plasma Treated PMMA Process 2
Contact Angle	80 degrees	20 degrees	14 degrees
Pull Strength	189.6 psi	523.1 psi	447.4 psi
Extension	0.064”	0.139”	0.148”
Failure Mode	Adhesion	Substrate	Substrate

Once treated, the material can be heated to form desired shapes. The heating time is dependent on the thickness of the sheet. TriStar offers full material surface modification as part of our machining services.

Machining PTFE

PTFE began as the brand name Teflon®, but filled versions are commonly referred to as Rulon®; a group of over 300 unique formulas. Rulon PTFE materials cover nearly every material type and industrial application, some formulas are abrasive, others offer NSA, FDA, and USP Class VI compliance, some serve in wet applications, others in dry conditions only. Rulon is a versatile material with unlimited design possibilities.

The key challenge in machining PTFE/Rulon is in holding the component without deforming it. Holding forces must be minimized and spread out over as much surface area as possible. It is often necessary to place mandrels into the IDs to reduce distortion.

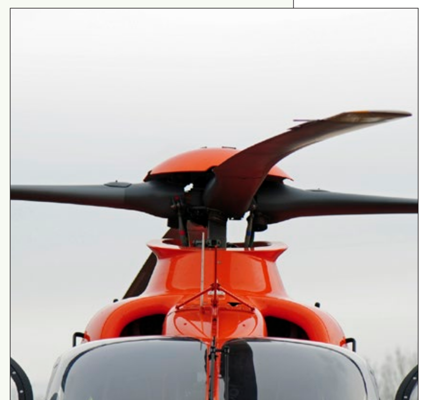
Unfilled or virgin PTFE can usually be machined using high-speed steel or high-polished carbide tooling without coolants; particularly if the cutting tools are sharp. Filled PTFE, however, is usually abrasive and requires carbide tooling. The material does respond well to coolants since it can easily resist water and is unaffected by most chemicals.

For screw machining, TriStar recommends a machining surface speed of 400 FPM. A feed rate of .004 to .006 in. per revolution is ideal when working with stock of 0.125” diameter or smaller; the bar will have to be supported during forming either through a center hole or on the side opposite the tool. Very small diameter pieces may require a much larger diameter ground rod to form the required diameter just prior to cut-off. Keep in mind that standard screw machines generally limit bar length to 12’.

Common applications of PTFE/Rulon:

- Medical equipment
- Aeronautical
- Meat processing
- Anemometers

The TriStar Advantage for Rulon/PTFE: Noise reduction, improved wear and performance



Challenge:

A leading maker of helicopter components approached us to improve the performance of their cowling rub strips, which were wearing out at an alarming rate. Located on the body panels (or skins) of the helicopters, the virgin Teflon rub strips could not withstand the constant friction of composite-on-composite that occurred over hundreds of takeoff and landing cycles. A custom solution was needed to increase wear, while reducing replacement and maintenance costs.

Solution:

Our Experts consulted with the design team, and recommended custom-machined Rulon J strips to install along the body panels. Using a surface speed of 400 RPM, TriStar machined the components. Rulon J has a PV rating of 7500, plus a unique filler for extended temperature stability. Rulon J has outperformed Teflon in this aerospace application. Our client also reports a significant reduction in composite friction and replacement rates from this durable custom material.

Machining PEEK

PEEK (Polyetheretherketone) is a high-temperature, semi-crystalline thermoplastic with superior chemical, wear and stability properties. PEEK has a working temperature range of 480°F but is able to resist temperatures over 550°F in steam and high-pressure environments. It offers very little moisture absorption. PEEK is available in standard shapes, but is an excellent candidate for machining.

PEEK standard sizes are sheet (0.03”-2.0” thick), rod (diameter 0.25”-6.0”) or tube. Common fillers include glass and carbon fibers. PEEK is a high-strength alternative to fluoropolymers. In some machine shops, PEEK materials have a dedicated set of tools, and fixturing devices to prevent contamination in parts to be used in the medical and food processing industries.

When machining PEEK, moderate cutting speeds and fast feed rates are generally called for. As far as tooling goes, polished carbide, diamond-filmed and PCD inserts work best. PCD cutters are the best choice for glass-filled PEEK. Using tools with small cutting radii will work without chopping the material or finish.

High cutting speeds with medium feed rates are recommended. Pay attention to minimize excess heat accumulation. Average SFPM for sawing is 2400, drilling 200-400, milling 400-800, and turning 800-1400.

Common applications of PEEK:

- Semiconductor machinery
- Aerospace components
- Pumps and valves
- Electric components
- Food processing

The TriStar Advantage for Machining PEEK: Reduced part distortion for higher production

Challenge:

A major manufacturer of food processing equipment required a material for their high-volume processing unit. The high temperatures on the unit were causing the positioning components to warp. To compensate for the high temperatures, a cooling unit was used, but the cooling time slowed down overall efficiency on the line.

Solution:

TriStar's engineering team paid a site visit, and custom fabricated components from high-temperature PEEK 1000. The PEEK material was able to withstand the high-temperatures of the machines, and eliminated the distortion it caused. The end result was improved production with fewer stoppages to address broken parts.

PEEK can be easily machined, and is available in various lengths, widths, thicknesses, and diameter tolerances. When purchased in rod form, we often attach the end of one end to another rod, to allow for a longer, continuous machining material. This technique allows for a higher yield with less scrap per rod.

Machining Composites

Composites are non-corrosive, low-maintenance and offer exceptional service life and excellent design versatility. Structural fibers vary based on application needs and may include carbon or glass. Ultracomp, our signature composite material, is non-toxic and produces no harmful odors or inhalants. Ultracomp is available in machined parts or plane bearings in the form of sleeve, flange and spherical bushings.

When machining Ultracomp, it is critical to review issues such as press fit dimensions and shaft clearances. Ultracomp also produces nearly as much dust as machining wood, with very fine particles. Graphite lubricated blends will also produce graphite powder in the dust, so an integrated collection system is critical.

Milling speeds and feeds are similar to turning. A bandsaw is recommended with metal, bi-metal, cobalt, or carbide blades for an optimal cut.

The TriStar Advantage for Machining Composites: Turnkey solution reduces delivery time

Challenge:

A major manufacturer of military technology systems contacted us to replace a metal rolling insert on a drum shaft. Their application posed a unique challenge since the drum is on a device that requires frequent and intensive wash downs since it is used to process waste products. Their metal bearings required constant maintenance at a prohibitive replacement cost. The client also complained that order fulfillment time — which averaged anywhere between four to five months — was unacceptable.

Solution:

Our in-house experts proposed Ultracomp 16" spherical bearings to replace the metal rolling element. Built of synthetic resins, Ultracomp bearings are exceptionally durable and excel in rotary applications. They also require no lubrication and can withstand frequent wash downs to help save on maintenance costs.

With TriStar's in-house machine shop, we were able to deliver a turnkey assembly for the client in just three weeks — remarkably better than the four-to-five months our client had waited for a metal spherical element.

Machining Plastics: Consider the benefits of outsourcing

While it is true that metal shops can machine plastics, there are often the issues of a steep learning curve and quality control. Once you've selected a material and conducted a performance-to-cost-analysis, in many instances, it is the expertise and creativity of the machining team that can make the difference between precision machining and poor machining. Look for a machine shop that not only invests in its equipment, but in operator training to help you make the most of your material investment.

At TriStar, our machining expertise has been built over 30 years across more than 70 industries. We have the material knowledge, machining skills and advanced equipment to bring your design ideas to reality.

Isn't it time to explore the TriStar Advantage for outsourced plastic machining?

Outsourcing your plastic machining can lead to reduced material cost, better-fitting components, and faster delivery. Experience the TriStar Advantage for precision machining.

TriStar

Engineered Plastic Solutions™

QUADRANT

This paper was prepared in association with Quadrant.

quadrantplastics.com

TriStar's machine shop features:

- Precision engineering- Manufactured to your specific requirements
- State-of-the-art fabrication facility- with CNC milling, turning and machining
- Rigid quality control and minimal lead-time – so that you receive the right material, the first time

TriStar's equipment inventory includes:

CNC Swiss Screw Machines

- High-speed turning
- Bar capacity up to 1.25"
- Continuous bar feeding
- 6-axis control system
- Secondary milling and drilling

CNC Milling

- Up to 36" x 81" travel
- Rapid tool change
- Close tolerance
- Prototype/production
- CAD/CAM

CNC Turning

- Live mill head attachments
- Bar capacity up to 2.75"
- Secondary milling & drilling
- Chucking capacity up to 21"

 Rev. 2 6/2015