Plastics Processing

Overview of Plastics Value-Added Services

Plastics Processing

CNC Panel Saw Cutting

Every Alro Plastics location has a CNC Saw for cutting plastics to the desired sizes. Our CNC production saws are capable of holding tight tolerances while cutting material up to 8" thick and 14 feet wide/long. With these heavy duty CNC saws we are able to cut thru a single block of 8" thick Nylon in one pass or we can stack multiple sheets of 1/4" thick Acrylic and cut them in one, efficient step. These high precision CNC saws are designed to fast and productive.

We pride ourselves on cutting and shipping the same day as ordered in most cases. With a few of our locations running two and even three shifts we are able to turnaround custom cut orders in a hurry. Along with our huge inventory of materials Alro Plastics can promise some of the fastest lead times in the industry. Give us a try on your next cut-to-size order and see what Alro can do for you!

Highlights

- Quantities: 1 pc - 50,000 pcs
- Thickness: 1/16" up to 8" thick
- Length/Width: 1/2" up to 168" wd/lg
- Standard Cut Tolerance: +1/16" / -0"
- Custom tolerances available by request
- Multiple shifts for shortest lead times

Plastics Processing

Rounds & Structural Cutting

Alro Plastics also offers same day cutting and shipping on our Rod and Tube stock. We stock up to 14" diameter in some materials and have the capabilities to cut up to 18" diameter in house. These saws can quickly cut a few pieces or be programmed to cut a production run. Our standard cut tolerance is always on the plus side so the pieces will never come in undersized. We also have the ability to cut U-channel, angle and various other profiles shapes on these saws.

Video available on

YouTube

We also drop tag and bin locate all of our cut off pieces to offer our customers smaller minimun orders. This is helpful to customers looking for just a few pieces for a proto-type run. Give your Alro representative the length you are looking for and they will check the drops to see if any will work for your job.

Highlights

- Quantities: 1 pc - 10,000 pcs
- Diameter: 1/8" up to 18" diameter
- Length/Width: 1" up to 20 feet long
- Standard Cut Tolerance: +1/4" / -0"
- Custom tolerances available by request
- Solid rounds and hollow rounds

Plastics Processing

Waterjet Cutting

With plastic sheets and slabs being offered in thicker sizes every year and the growing number of materials that are challenging to cut with traditional methods, Alro Plastics offers Waterjet cutting. The Waterjet is capable of cutting sheets up to 10" thick in a single pass to precise tolerances. This capability allows us to offer cut-to-size pieces beyond what our CNC saws can perform.

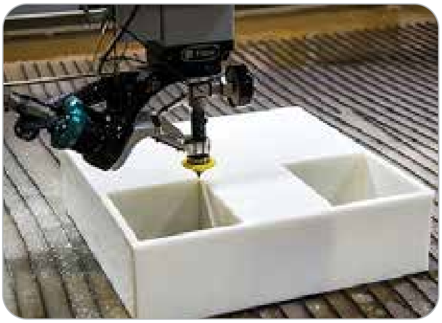
Video available on YouTube

Another advantage to Waterjet cutting is the ability to cut a wide range of challenging materials with ease. Materials like fiberglass, G10, glass-filled plastics, urethane, neoprene, rubber and even foam can be cut on the Waterjet. These materials were difficult to cut in the past but with the Waterjet Alro Plastics is able to offer cut-to-size pieces in all these materials.

This 5-axis machine can also do bevels and 3D cutting as well as "per print" cutting. We can upload drawings and prints to the Waterjet and cut out an assortment of shapes and contours as well as cut out holes and other shapes. This versatile machine allows us to offer even more unique services to our customers.

Highlights

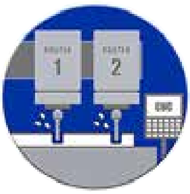
- Quantities: 1 pc - 20,000 pcs
- Thickness: 1/32" up to 10" thick
- Width: 1/2" up to 78" wide
- Length: 1/2" up to 157" long
- Standard Cut Tolerance: +/- .015"
- Custom tolerances available by request
- Fiberglass, G10, glass-filled materials, rubber, urethane, foam and more



Plastics Processing

CNC Production Routers

Alro has multiple CNC Routers to machine custom plastic parts per print. These machines are capable of extremely close tolerances for milled, drilled and routed parts up to 120" wide x 144" long and up to 4" thick can be achieved. Arcs, curves, slotted tracks, machine guards and conveyor parts are just a handful of the many parts we are capable of producing. E-mail us your drawings at plastics@alro.com, we accept .dwg, .dxf, and .iges files.



We take pride in being ISO certified and do everything in our power to meet the most demanding lead times. Our goal is to keep our lead times on CNC routered parts to 2 weeks or less from stock material. Give us a shot on your next fabricated plastic part and see if we can exceed your expectations.

Highlights

- Quantities: 1 pc - 50,000 pcs
- Thickness: 1/16" up to 4" thick
- Width: 1/2" up to 120" wide
- Length: 1/2" up to 144" long
- Standard Cut Tolerance: +/- .015"
- Custom tolerances available by request
- Multi-Table Routers for production runs
- Automatic tool changers, up to 32 tools

Plastics Processing

Milling & Machining

Alro Plastics utilizes Haas VMC (Vertical Machining Centers) to give us the ability to machine small, intricate parts per print. The VMCs are able to machine parts from 1/16” up to 6” thick. It also has a 32” wide x 60” long table surface to hold a wide variety of parts. These machines are also great for secondary machining work and compliment our CNC Routers nicely.

Video available on YouTube

With its automatic tool change and ease of programming, we are able to machine complicated parts to print in a single operation. The rapid travel of the tool change process shaves time off the process enabling us to reduce run time and turn our more parts per cycle. The ease of programming allows us to quickly set up small run jobs and prototype runs with minimal cost and lead time.

Highlights

- Quantities: 1 pc - 10,000 pcs
- Thickness: 1/16" up to 6" thick
- Width: 1" up to 32" wide
- Length: 1" up to 60" long
- Ideal for prototypes and small, complex parts
- Custom tolerances available by request

Plastics Processing

Drilling & Tapping

Our FlexArm equipment allows Alro Plastics to offer in house capability of part tapping and Helicoil inserts. The FlexArm keeps the Helicoil insertion tool perpendicular to the work piece. The depth control ability offers consistency from part to part whether tapping or inserting helicoils. A FlexArm Tapping Machine will take care of prep work such as reaming, chamfering and deburring.

Highlights

- Quantities: 1 pc - 10,000 pcs
- Thickness: 1/4" up to 4" thick
- Length/Width: 1" up to 120" wide/long
- Helicoil Sizes:
- Standard: 8-32, 10-24, 10-32, 1/4-20, 3/8-20 and 1/2-13
- Metric: M3-0.5, M4-0.7, M5-0.8, M6 x 1.0, M8 x 1.25 and M10 x 1.50

Plastics Processing

Plastic Welding

When most people hear the word welding they immediately think of steel and metal welding. Well, did you know that some plastics can also be welded? Polypropylene, polyethylene, PVC, PVDF, ABS and certain thermoplastics can all be welded.

Alro Plastics has invested the time and resources to become very good at plastic welding and we offer this service to all of our customers. From the simple task of butt-welding two sheets together to make one longer sheet, to the complex process of creating custom fabricated tanks to print, we can do it all.

Highlights

- Hot gas and modified extrustion welding
- Done in house, better control of lead times
- Pieces machined on routers for best finish
- Experienced welders specializing in plastics
- Many plastics can be welded

Plastics Processing

Bend, Glue & Polish

Alro Plastics also offers our customers custom fabricated bent and glued parts to print. Some of our thinner gage plastics can be heat bent or cold formed on a press break, mostly our “See-through” materials like Acrylic, Plexiglas, Polycarbonate and PETG.

We can also CNC Saw Cut and Router these parts and assemble them together on custom fabricated jobs. ISO certified to ensure high quality finished parts in a timely manner. Please email us at plastics@alro.com for a quote!

Highlights

- CNC machined edges for the best bonds
- Ability to heat bend plastics per print
- Cold forming and bending also available
- Experienced fabricators specializing in plastics
- Complete assembly of finished parts

Machining Guidelines

For their mechanical, chemical, and lightweight properties, plastics are increasingly replacing metals in a wide range of engineering applications. Many of the same machining methods used to fabricate metal parts are also used for plastics; however, best practices for machining plastics differ considerably. Plastics exhibit a number of properties that influence machining procedures, including:

- Thermal expansion is up to 10 times greater with plastics than metals
- Plastics lose heat more slowly than metals, so avoid localized overheating
- Softening (and melting) temperatures of plastics are much lower than metals
- Plastics are much more elastic than metals

From material selection to proper tooling, from feed rates to stabilizing methods, part producers must weigh a range of factors in order to achieve good results when machining plastics. The following guidelines cover the most common plastic machining methods and provide useful tips and data for working with engineering polymers from Mitsubishi Chemical Group.

Plastic machining processs

This section covers the most common methods of machining plastic engineering components, providing guidelines and tips to achieve the best results with each. Proper machining is crucial to achieving part dimensions and performance. Improper machining can create stress within the finished part, negatively impacting its mechanical properties and risking premature part failure.

Threading and Tapping

What is threading? What is tapping?

Tapping and threading are two machining methods used to produce screw threads. Threading is the process of using a die tool to carve external threads, where tapping is the process of using a tap tool to create threads on the inside of a drilled hole.

Threading and tapping with plastic vs. metal

A primary consideration when threading and tapping plastic is that plastics are more notch-sensitive than metals. Some polymer materials may tear during threading, especially in fine pitch procedures.

Threading tips for plastic

Threading plastic should be done by single point using a carbide insert, taking four to five 0.001” passes at the end. Coolant usage is suggested.

Tapping tips for plastic

When tapping plastic, use the specified drill with a two-flute tap. Keep the tap clean of chip build-up. Use of a coolant during tapping is also suggested.

Machining Guidelines

Milling

What is milling?

Milling is a machining method that applies a high-speed cylindrical cutting tool to a stationary plastic shape, moving the cutter on an axis to subtract from the shape in different directions. Computerized numerical control (CNC) milling increases the accuracy and efficiency of plastic milling.

Milling plastic vs. metal

When milling plastic, it is crucial to properly stabilize the part on the worktable and minimize vibrations from the high-speed cutting tool – these may result in chatter marks and decreased accuracy due to the shape wandering.

Milling tips for plastic

- Climb milling, also known as down milling, is recommended over conventional milling.
- Sufficient fixturing on the mill bed allows fast table travel and high spindle speeds.
- The shape should not be fixed too tightly, however, as it may deform or spring.
- When face milling, use positive geometry cutter bodies.

Sawing

Sawing is a machining method that involves cutting a material into multiple pieces using a bandsaw, table saw, or other specialized equipment.

Sawing plastic vs metal. The primary difference between sawing plastic vs. metal is that the heat generated by the saw blade can negatively impact the plastic parts due to lower softening and melting temperatures. It is crucial to account for the speed of the blade, the thermal properties of the material, and the thickness of the shape when sawing plastic.

Tips for selecting a saw tool

- Band saws are versatile and perform well for straight, continuous curves, and irregular cuts.
- Table saws are convenient for straight cuts and can be used to cut multiple thicknesses and thicker cross sections – up to 4” with adequate horsepower.

Tips for selecting a saw blade

- Rip and combination blades with a 0° tooth rake and 3° to 10° tooth set are best for general sawing in order 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 optimum surface finishes.

Machining Guidelines

Drilling and Boring

What is drilling? What is boring?

Drilling is a machining method that creates cylindrical holes and throughholes by means of a pointed helical tool. Boring is a secondary process for enlarging or finishing drilled holes.

Drilling and boring plastic vs. metal

The heat insulating characteristics of plastics require consideration during drilling operations, especially when the depth of the hole is greater than twice its diameter. Excessive heat build-up may result in chipping, rough surfaces, and inadequate tolerances.

Drilling tips for plastic

For small diameter holes (1/32" to 1" dia.)

- High-speed steel twist drills are generally sufficient.
- Frequent pullout (peck drilling) is suggested to improve swarf removal.
- A slow spiral (low helix) drill will allow for better swarf removal.

For large diameter holes (1" and greater)

- A slow spiral (low helix) drill or general-purpose drill bit ground to a 118° point angle with 9° to 15° lip clearance is recommended. The lip rake should be ground (dubbed off) and the web thinned.
- Avoid hand feeding - drill grabbing can result in microcracks.
- It is generally best to drill a pilot hole (maximum 1/2"dia.) using 600 to 1,000 rpm and positive feed of 0.005" to 0.015" per revolution.
- Secondary drilling at 400 to 500 rpm at 0.008" to 0.020" per revolution is required to expand the hole to larger diameters.

For especially notch-sensitive materials (such as Ertalyte® PET-P and glass reinforced materials)

- A two-step process involving both drilling and boring minimizes heat build-up and reduces the risk of cracking.
- First, drill a 1" diameter hole using an insert drill at 500 to 800 rpm with a feed rate of 0.005" to 0.015" per revolution.
- Next, bore the hole to final dimensions using a boring bar with carbide insert with 0.015" to 0.030" radii at 500 to 1,000 rpm and feed rate of 0.005" to 0.010" per revolution.

MITSUBISHI CHEMICAL GROUP

Machining Guidelines

Reaming

Most of the engineering plastics can be reamed with either hand or collar reamers to produce holes with good finish and accurate dimensions. Expansion type reamers and standard .001-.002" oversize stub machine reamers can also be used. Helical flute reamers are recommended if there is an interruption in the I.D.. Cuts made with a fixed reamer tend to be undersized unless at least .005" is removed by the final reaming. With a .01-.02" per revolution feed rate and a .005-.01" depth of cut, reamer speeds of 250-450 fpm are recommended.

Reaming PTFE / TFE is generally not recommended. The operation causes the material to compress, especially if the reamer is not exceptionally sharp. Also, PTFE's elasticity will cause holes to "fall in" creating undersize holes. If necessary, special reamers with a primary relief (clearance) angle can produce accurate holes. The use of an oversized reamer can correct undersized holes. Where hole diameter permits, a single point boring tool is recommended to finish the hole to close tolerances.

Turning

What is turning?

Turning is a machining process in which a plastic shape is rotated around a stationary lathe. Turning is especially useful for machining parts that are symmetrical along a common rotational axis.

Turning plastic vs. metal

As with other plastic machining processes, turning generates heat. In order to prevent damage to a plastic part, rotation speed, tool selection, and coolants should all be considered carefully along with the thermal properties of the material.

Turning tips for plastic

- Turning operations require inserts with positive geometries and ground peripheries.
- Ground peripheries and polished top surfaces generally reduce material build-up on the insert, improving the attainable surface finish.
- A fine-grained C-2 carbide is often recommended for plastic turning operations.

Machining Guidelines

Coolants

Coolants are generally not required for most machining operations (not including drilling and parting off). However, for optimum surface finishes and close tolerances, non-aromatic, water soluble coolants are suggested. Spray mists and pressurized air are very effective means of cooling and cutting interface. General purpose petroleum based cutting fluids, although suitable for many metals and plastics, may contribute to stress cracking of amorphous plastics such as PC 1000 Polycarbonate, PSU 1000 Polysulfone, Ultem® 1000 PEI, and Radel® R PPSU.

Coolant tips for plastics

- Coolants are strongly suggested during drilling operations, especially with notch sensitive materials such as Ertalyte® PET-P, Torlon® PAI, Celazole® PBI and glass or carbon reinforced products.
- In addition to minimizing localized heat-up, coolants prolong tool life. Two (flood) coolants suitable for most plastics are Trim 9106CS (Master Chemical Corp. - Perrysburg, OH) and Polycut (Tullco -Savannah, GA). A generally suitable mist coolant is Astro-Mist 2001A (Monroe Fluid Technology - Hilton, NY).

Post Annealing

If during the machining process significant material is removed, annealing is recommended to relieve machined-in-stress and minimize possibility of premature part failure.

Acrylic parts should be annealed at 175°F only after fabrication and polishing are completed. Anneal 10 hours for thicknesses up to .150", an additional 30 minutes for each 1/4" up to 1-1/2", and an additional hour for each 1/4" thereafter. Parts should be supported while annealing takes place in an air medium. Slow cooling is also highly recommended.

Delrin® should be air annealed at 320°F for 30 minutes plus 5 minutes per 0.04" of wall thickness. It is important that the parts be uniformly heated and the oven capable of controlling the circulating air temperature to ±5°F. Oil annealing in an oil circulating bath at 320°F will require 15-20 minutes to come up to temperature plus 5 minutes per 0.04" of wall thickness. Uniform heating is important, and the parts should be restrained from contact with each other and the walls of the bath.

Nylon should be annealed in the absence of air, preferably by immersion in a suitable liquid. A temperature of 300°F is often used for general annealing. Annealing time is 15 minutes per 1/8" of cross-section. When removed from the bath, the material should be allowed to cool in the absence of drafts. The choice liquid to be used as the heat transfer medium should be based on the following considerations:

- Heat range and stability should be adequate
- Should not attack Nylon
- Should not give off noxious fumes or vapors
- Should not present a fire hazard
- High boiling hydrocarbons, such as oils and waxes, may be used

Trouble Shooting - Drilling

DIFFICULTY	COMMON CAUSE
Tapered Hole	1. Incorrectly sharpened drill 2. Insufficient clearance 3. Feed to heavy
Burnt or Melted Surface	1. Wrong type drill 2. Incorrectly sharpened drill 3. Feed to light 4. Dull drill 5. Web too thick
Chipping of Surfaces	1. Feed to heavy 2. Clearance too great 3. Too much rake (thin web as descibed)
Chatter	1. Too much clearance 2. Feed light 3. Drill overhang too great 4. Too much rake (thin web as descibed)
Feed marks or Spiral lines on Inside Diameter	1. Feed too heavy 2. Drill not centered 3. Drill ground off-center
Oversize Holes	1. Drill ground off-center 2. Web too thick 3. Insufficient clearance 4. Feed rate to heavy 5. Point angle too great
Undersize Holes	1. Dull drill 2. Too much clearance 3. Point angle too small
Holes Not Concentric	1. Feed to heavy 2. Spindle speed to slow 3. Drill enters next piece too far 4. Cut-off tool leaves nib, which deflects drill 5. Web too thick 6. Drill speed to heavy at start 7. Drill not mounted on center 8. Drill not sharpened correctly
Burr at Cut-off	1. Dull cut-off tool 2. Drill does not pass completely through piece
Rapid Dulling of Drill	1. Feed too light of drill 2. Spindle speed to fast 3. Insufficient lubrication from coolant

Trouble Shooting - Turning & Boring

DIFFICULTY	COMMON CAUSE
Melted Surface	1. Tool dull or heel rubbing 2. Insufficient side clearance 3. Feed rate to slow 4. Spindle speed to fast
Rough Finish	1. Feed to heavy 2. Incorrect clearance angles 3. Sharp point on tool (slight nose radius required) 4. Tool not mounted on center
Burrs at Edge of Cut	1. No chamfer provided at sharp corners 2. Dull tool 3. Insufficient side clearance 4. Lead angle not provided on tool (tool should ease out of cut gradually, not suddenly)
Cracking or Chipping of Corners	1. Too much positive rake on tool 2. Tool not eased into cut (tool suddenly hits work) 3. Dull tool 4. Tool mounted below center 5. Sharp point on tool (slight nose radius required)
Chatter	1. Too much nose radius on tool 2. Tool not mounted solidly 3. Material not supported properly 4. Width of cut too wide (use 2 cuts)

Trouble Shooting - Cutting Off

DIFFICULTY	COMMON CAUSE
Melted Surface	1. Dull tool 2. Insufficient side clearance 3. Insufficient coolant supply
Rough Finish	1. Feed to heavy 2. Tool improperly sharpened 3. Cutting edge not honed
Spiral Marks	1. Tool rubs during its retreat 2. Burr on point of tool
Concave or Convex Surfaces	1. Point angle to great 2. Tool not perpendicular to spindle 3. Tool deflecting 4. Feed to heavy 5. Tool mounted above/below center
Nibs or Burrs at Cut-off Point	1. Point angle not great enough 2. Tool dull 3. Feed too heavy
Burrs on Outside Diameter	1. No chamber before cut-off diameter 2. Dull tool

Saw Cutting - Kerf Line

Whenever you cut up a full sheet of material into smaller pieces you lose some material due to the thickness of the blade. When the saw enters the material it chews up the material it is cutting through. This is referred to as the "Kerf Line" or wasted material. In order to properly yield out the correct amount of pieces one can expect from a full sheet you need to know how much Kerf to account for. The majority of our blades for our CNC Saws are .187"thick. You also need to factor in the cut tolerance as well, our standard cut tolerance is +1/16" / -0". With tolerances you want to shoot for the middle of the tolerance, so you have some flexibility either way.

Saw Blade (.187"thk) + Tolerance (.063") = .250" KERF Line.

We would then take the Kerf Line and add that number to each of our cut dimensions. For example, say our finished size is 7" x 11", and we will be cutting these out of 48" x 96" full sheets. So if you take the cut sizes and add the Kerf line you can then properly figure out the best possible yield from a full sheet of material.

7.00" + .250" = 7.250" and 11.00" + .250" = 11.250"

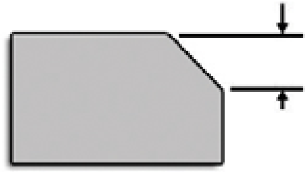
Now we take our full sheet sizes and divide them by the cut pieces with Kerf factored in to figure out the best possible yield.

48" / 7.250" = 6 pcs and 96" / 11.250" = 8 pcs. Full sheet yield = 48 pcs
48" / 11.250" = 4 pcs and 96" / 7.250" = 13 pcs. Full sheet yield = 52 pcs.

So the best yield is the second example, we could get 52 pcs from one full sheet of material if we cut it this way. If the total job was for 200 pcs, then we would need 4 sheets of material in order to get the 200+ pcs required. This is the same method we use when we are quoting our customers on any cut-to-size job.

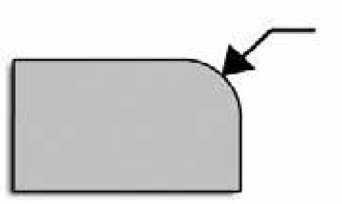
The "kerf" line can be seen in the above photo. Notice the faint line following the blade, that is the kerf line, where the blade cut through the sheet of plastic material.

Standard Edge Finishes



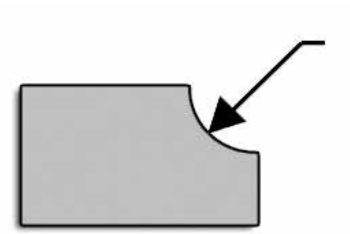
CHAMFERED EDGE CUT

Generally called out as 1/8" x 45 degrees or 1/8" x 1/8". The typical chamfer cut uses a 45 degree bit, but some prints do call out different degrees. The chamfer dimension is generally called out from the surface of the material down to the specific size.



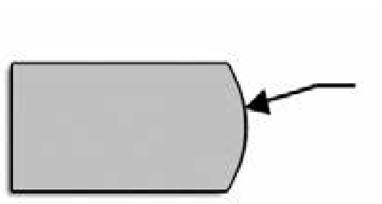
ROUND OVER EDGE

Usually used to break a sharp edge leftover from machining the part. By going back thru and rounding over the edges it reduces the chance of getting your hand cut or sliced. Ideal for work surfaces or some machine guards. Generally called out by the size of the Radius needed, ex. 1/8" Radius.



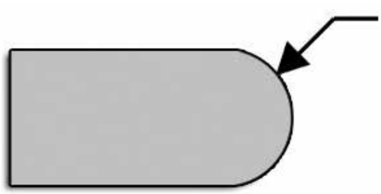
REVERSE RADIUS EDGE

Not as common as the others, more for decoration than anything else. Basically just come by with a ball-end cutter and drop it down to the desired depth. May have seen these on a picture frame or something similar.



FINGERNAIL EDGE CUT

Similar to the Full Radius (below), it is a little less round. The radius is always larger than half the thickness of the material. So the radius does not blend into the thickness smoothly, it ends abruptly. Still leaves a nice rounded edge to reduce any sharp corners. This cut is a lot easier to do than the Full Radius.



FULL RADIUSED EDGE

In the case of the Full Radius, the radius is exactly half the size of the thickness of the material or the diameter and thickness are the same size. This allows the radius to flow perfectly and smoothly into the thickness of the material. This is a very hard cut for most Plastics, since the thickness of the material can vary as much as +/-10% on some Plastics.