

GUIDE TO MACHINING PLASTIC PARTS



CONTENTS

Stock Shapes

Machining instructions for stock shapes 3

Cooling

Cooling plastics 4

Sawing

Cutting tools 5

Data sawing 6

Troubleshooting sawing 6

Drilling

Drilling 7

Data drilling 8

Troubleshooting drilling 9

Turning

Tapping/Turning threads 10

Turning tips 11

Troubleshooting turning 11

Data turning 12

Milling

Milling 13

Data milling 14

Tolerances

Best tolerances 15

Engineered solutions 15

Thread inserts /Deburring

Thread inserts 16

Deburring 16

General rules

General rules 17

Solution example 17

Annealing

Better flatness and tighter tolerances 18

Annealing tips 18

Recommended annealing procedure 18

Summary

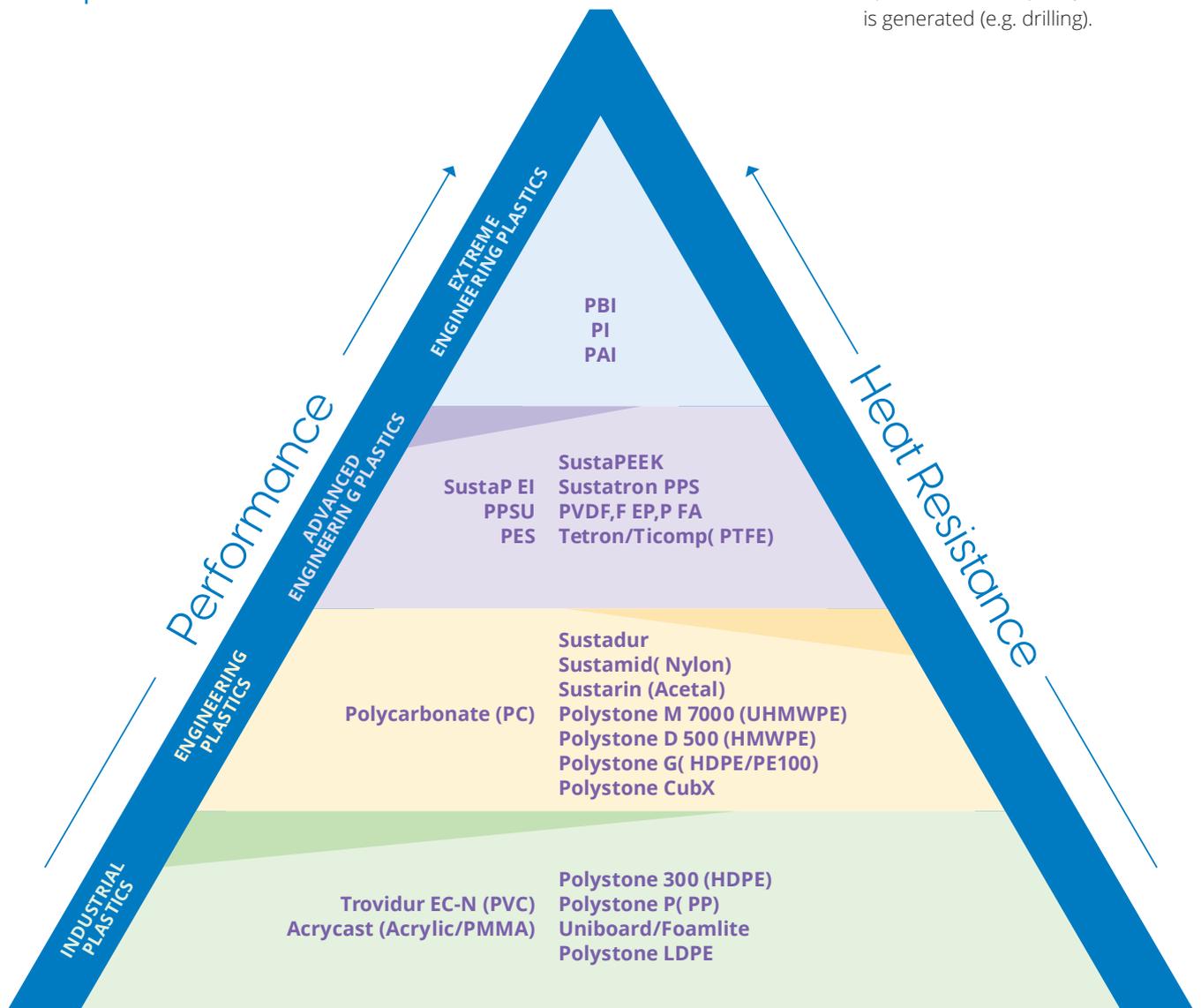
Summary 19

MACHINING INSTRUCTIONS FOR STOCK SHAPES

Stock shapes can easily be machined on ordinary metalworking and in some cases on woodworking machines. However, there are some points which are worth noting to obtain improved results.

Tools must be always kept sharp and smooth; specific polymer group determines the type of tools to be used.

- Feed rates should be as high as possible.
- Tools must have sufficient clearance so that the cutting edge only comes in contact with the plastics material.
- A good swarf removal from the tool must be assured.
- Coolants should be applied for operations where plenty of heat is generated (e.g. drilling).



KEEP IT COOL!

Due to poor thermal conductivity and lower melting properties the main skill is to avoid heat build up and evacuate swarf. This avoids deformation, stress build up or even melting.

- When machining Polyamides, UHMW-PE and softer materials go for maximum feed and speed to get swarf away from tool and avoid swarf build up or wrap around.
- Select tools with large clearance to allow swarf evacuation and deeper cuts.
- Use milling cutters with two or three flute with higher helix.
- Try to use air blast or vacuum venturi.
- HSS works well but solid carbide is stiffer.
- Try to use air blast.
- On glass or carbon fiber use carbide, reduce speed and increase feed rates.



COOLING PLASTICS

Coolants are generally not required for most machining operations (not including drilling and parting off). However, for optimum surface finishes and close tolerances, nonaromatic, water-soluble coolants are suggested. Spray mists and pressurized air are very effective means of cooling the cutting interface. Mineral oil based cutting

fluids, although suitable for many metals and plastics, may contribute to stress cracking of amorphous plastics such as Palsun PC, PPSU, SustaPEI and Sustason PSU. If using coolant on amorphous materials wash immediately in Isopropyl alcohol and pure water post machining!



CUTTING TOOLS

The cutting (polymer) material naturally has a serious effect on the service life of the tool. It's recommended to use HSS, carbide, diamond coated, PCD or CVD cutting tools.

- HSS high-speed steel tools work well with many plastics.
- Carbide tools are preferred for longer production runs.
- Polycrystalline diamond (PCD) or chemical vapour deposition (CVD) diamond tip tooling are essential when machining glass and/ or carbon fiber reinforced or graphite filled materials.
- Machining PBI, or PI polycrystalline diamond tooling provides optimum results, carbide tipped tools can then be used in case of very short production runs.

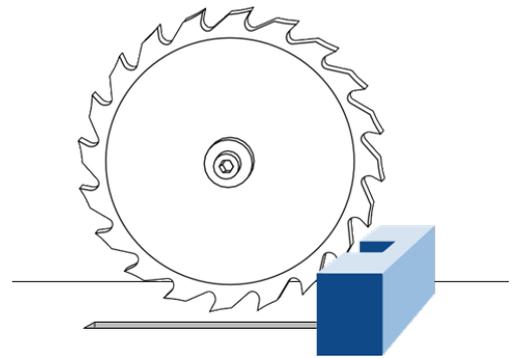
BILLETING/CUTTING

- Circular, band, reciprocating and guillotine saws are generally suitable for cutting thermoplastics.
- Set of teeth (clearance between teeth side and body of cutter) must be as large as possible to prevent material closing and trapping blade.
- Teeth per inch (TPI) must be suited for material:
 - soft low temperature materials need low TPI
 - filled and thinner materials require higher TPI.

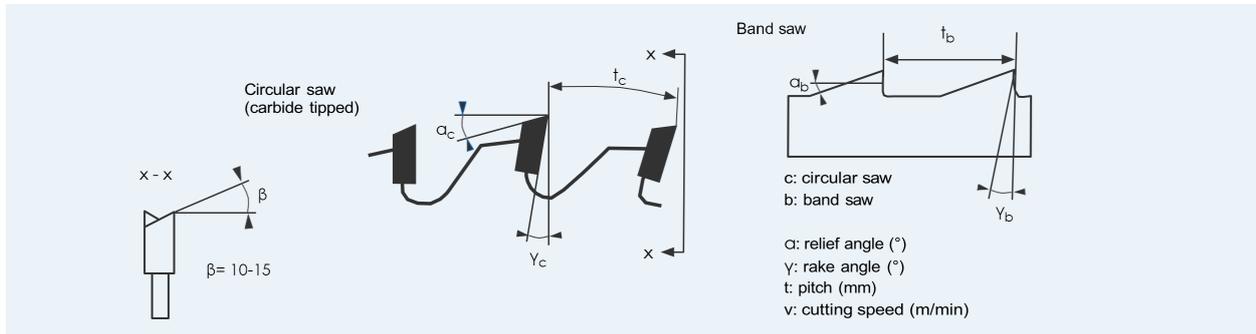
Reinforced materials such as Sustamid 66 GF 30, PAI, SustaPEEK MOD, SustaPEEK GF 30, SustaPEEK CF 30, and Sustatron PPS MOD are preferably cut with a band saw with a tooth pitch of 4 to 6 mm. Do not use standard circular saws as this usually leads to cracks!

CLAMPING TOOLS

- Circular, band, reciprocating and guillotine saws are generally suitable for cutting thermoplastics.
- Set of teeth (clearance between teeth side and body of cutter) must be as large as possible to prevent material closing and trapping blade.
- Teeth per inch (TPI) must be suited for material:
 - soft low temperature materials need low TPI
 - filled and thinner materials require higher TPI.



DATE SAWING



	α_c	γ_c	t_c	v_c	α_b	γ_b	t_b	v_b
Sustamid PA / Sustaglide PA, Polystone PE	10 - 15	0 - 15	8 - 45	1,000 - 3,000	25 - 40	0 - 8	4 - 10	50 - 500
Sustarin POM	10 - 15	0 - 15	8 - 45	1,000 - 3,000	25 - 40	0 - 8	4 - 10	50 - 500
Sustadur PET, PAI, SustaPEEK	10 - 15	0 - 15	8 - 25	1,000 - 3,000	25 - 40	0 - 8	4 - 10	50 - 400
Palsun PC, SustaPEI, Sustason PSU	10 - 15	0 - 15	8 - 25	1,000 - 3,000	25 - 40	0 - 8	4 - 6	50 - 400
PAI, SustaPEEK GF 30 / MOD / CF 30, Sustatron PPS MOD,	10 - 15	0 - 15	8 - 25	1,000 - 3,000	25 - 40	0 - 8	4 - 6	50 - 200
PBI, PI	10 - 15	0 - 15	8 - 25	1,000 - 3,000	25 - 40	0 - 8	2 - 3	25 - 100
Tetron/Ticomp/Tetco PTFE	10 - 15	0 - 15	8 - 25	1,000 - 3,000	25 - 40	0 - 8	4 - 6	50 - 200

TROUBLESHOOTING SAWING

DIFFICULTY	MELTED SURFACE	ROUGH FINISH	BURRS AT EDGE OF CUT	CRACKING OF CHIPPING OF CORNERS	CHATTER
Common cause	<ul style="list-style-type: none"> • Tool dull or heel rubbing • Insufficient side clearance • Feed rate too slow • Spindle speed too fast 	<ul style="list-style-type: none"> • Feed too heavy • Incorrect clearance angles • Sharp point on tool (slight nose radius required) • Tool not mounted on center 	<ul style="list-style-type: none"> • No chamfer provided at sharp corners • Dull tool • Insufficient side clearance • Lead angle not provided on tool (tool should ease out of cut gradually, not suddenly) 	<ul style="list-style-type: none"> • Too much positive rake on tool • Tool not eased into cut (tool suddenly hits work) • Dull tool • Tool mounted below center 	<ul style="list-style-type: none"> • Too much nose radius on tool • Tool not mounted solidly • Material not supported properly • Depth of cut too heavy (use 2 cuts)

DRILLING

Plastics can build up heat very easily during drilling operations, especially when hole depths are greater than twice the diameter. Therefore a cooling liquid is generally recommended.

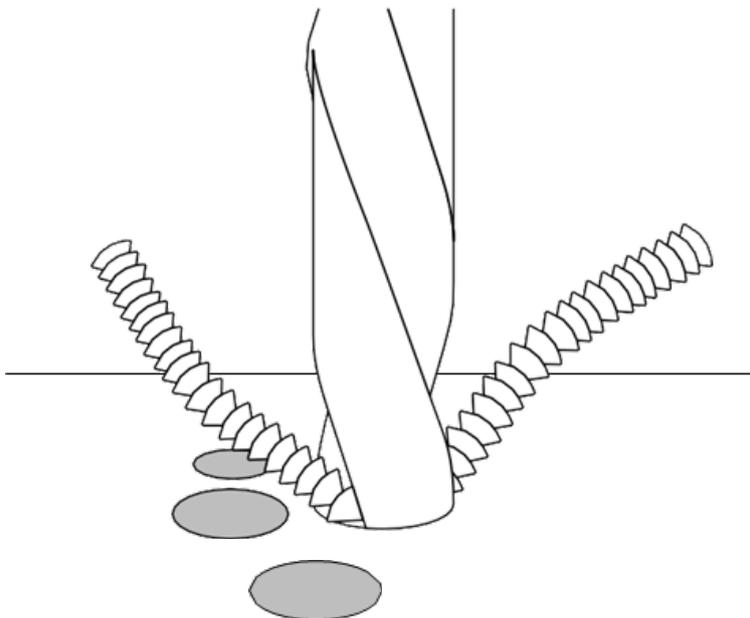
Small diameter holes (0.5 – 25 mm diameter)

High speed steel twist drills generally work well. In order to improve heat and swarf removal, frequent pull-outs (peck-drilling) are necessary. A slow spiral (low helix) drill will allow for better swarf removal.

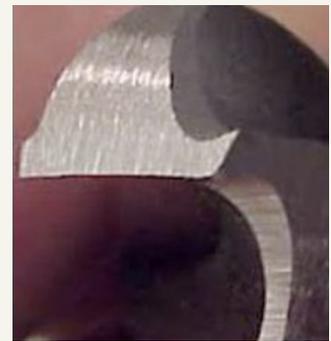
Large diameter holes (25 mm diameter and larger)

It is advised to use drills with a thinned web (dubbed drill) in order to reduce friction and hence heat generation. Drill large holes stepwise: a bore diameter of 50 mm e.g. should be made by drilling successively with \varnothing 12 mm and \varnothing 25 mm, then expanding the hole further with larger diameter drills with a single boring tool.

For pilot holes for internal threads always check actual cutting size of drill. Most plastics close up after drilling. This can make taps tight.



Thinning of web is recommended on larger drills



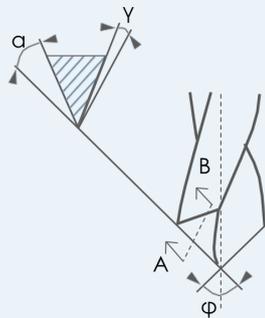
Insert drill on solid pieces with right inserts



Spade drills can be useful

DATA DRILLING

Section AB



ϕ : top angle (°)
 α : relief angle (°)
 γ : rake angle (°)
 s : feed (mm/rev.)
 v : cutting speed (m/min)

	α	γ	ϕ	s	v	MATERIAL TOOL
Sustamid PA / Sustaglide PA, Polystone PE	5 - 10	3 - 5	90 - 120	0.1 - 0.3	50 - 100	HSS steel or carbide
Sustarin POM	5 - 10	3 - 5	90 - 120	0.1 - 0.3	50 - 100	HSS steel or carbide
Sustadur PET, PAI, SustaPEEK	5 - 10	3 - 5	90 - 120	0.1 - 0.3	50 - 80	Carbide
Palsun PC, SustaPEI, Sustason PSU	5 - 10	3 - 5	90 - 120	0.1 - 0.3	50 - 100	Carbide
PAI, SustaPEEK GF 30 / MOD / CF 30, Sustatron PPS MOD,	5 - 10	3 - 5	90 - 120	0.1 - 0.3	50 - 80	Carbide or PCD/CVD
PBI, PI	5 - 10	3 - 5	90 - 120	0.1 - 0.3	25 - 50	Carbide or PCD/CVD
Tetron/Ticomp/Tetco PTFE	5 - 10	3 - 5	90 - 120	0.1 - 0.3	50 - 100	Carbide

TROUBLESHOOTING DRILLING

DIFFICULTY	TAPERED HOLE	BURNED OR MELTED SURFACE	CHIPPING OF SURFACES	CHATTER	FEED MARKS OR SPIRAL LINES ON INSIDE DIAMETER
Common Cause	<ul style="list-style-type: none"> • Incorrectly sharpened oil • Insufficient clearance • Feed too heavy 	<ul style="list-style-type: none"> • Wrong type drill • Incorrectly sharpened oil • Feed too light • Web too thick 	<ul style="list-style-type: none"> • Feed too heavy • Clearance too great • Too much rake (thin web as described) 	<ul style="list-style-type: none"> • Too much clearance • Feed too light • Drill overhang too great • Too much rake (thin web as described) 	<ul style="list-style-type: none"> • Feed too heavy • Drill not centered • Drill ground off-center
	OVERSIZE HOLES	UNDERSIZE HOLES	HOLES NOT CONCENTRIC	BURR AT CUT-OFF	RAPID DULLING OF DRILL
	<ul style="list-style-type: none"> • Drill ground off-center • Web too thick • Insufficient clearance • Feed rate too heavy • Point angle too great 	<ul style="list-style-type: none"> • Dull drill • Too much clearance • Point angle too small 	<ul style="list-style-type: none"> • Feed too heavy • Spindle speed too slow • Drill-off too leaves nib which deflects drill • Web too thick • Drill speed too heavy at start • Drill not mounted on center • Drill not sharpened correctly 	<ul style="list-style-type: none"> • Dull cut-off tool • Drill does not pass completely through piece 	<ul style="list-style-type: none"> • Feed too light • Spindle speed too fast • Insufficient lubrication from coolant

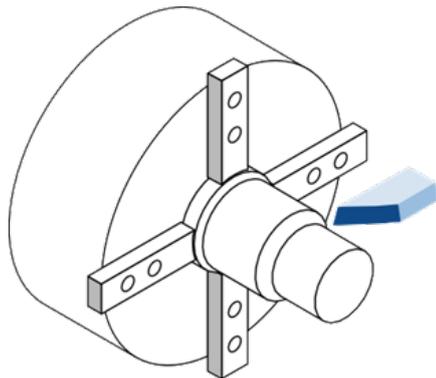
TAPPING/TURNING THREADS

Grades for grooving inserts

Stellram grades SP4030 and SP0436 for grooving inserts are PVD-coated grades with a micrograin substrate. They are designed for machining at higher speeds and lower feeds. Their grades are ideal for use on stainless steels and ductile materials. See inch and metric grade descriptions.

Ultra-Mini internal grooving tools

Our Ultra-Mini grooving inserts and tool holders portfolio features replaceable carbide inserts with minimum bores down to 0.157" and groove widths from 0.039" to 0.078". The inserts are designed for internal machining of small diameters and are available in a range of reaches and groove width-to-length ratios. See the full inch and metric Ultra-Mini tool offering. Also see inch and metric Mini-Cut for internal grooving.



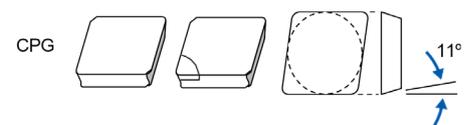
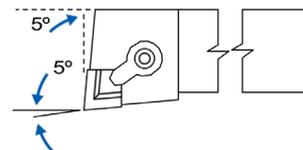
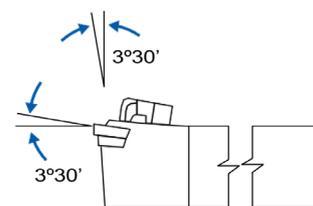
- Use of spiral taps is recommended (not straight).
- Where possible use rigid tap CNC cycles.
- Check pilot drill size.
- Go for high quality taps where possible.
- Turning threads should be straight forward.
- Tools do not have to be as rigid as metal cutting tools.



TURNING

Tip geometry

The continuous chip stream produced when turning and boring many thermoplastics can be handled well using a compressed air powered suction system (directly disposing the swarf into a container), in this way avoiding the chip wrapping around the chunk, the tool or the workpiece.



TURNING TIPS

- Machines should have cavernous area for swarf.
- Avoid conveyor belts.
- For tubes you can bore one end and put a blank in to support clamp force of jaws.
- Consider fixed pressure jaws on new machines.
- Crawford collets work very good for plastic.
- Use tip designed for non-ferrous metals such as aluminum or brass.
- Best finish is achieved with 0.8 rad on tips.
- Use hand diamond tools to polish grind marks.
- Sometimes it is worth grinding insert tips on top face to sharpen front edge.
- For improved tolerances, rough out large sections and leave for 24 hours.
- Most diameters will grow so aim for lowest dimension when setting.
- Consider doing capability studies on your machines.
- Use sliding head for small long sections.

TROUBLESHOOTING TURNING

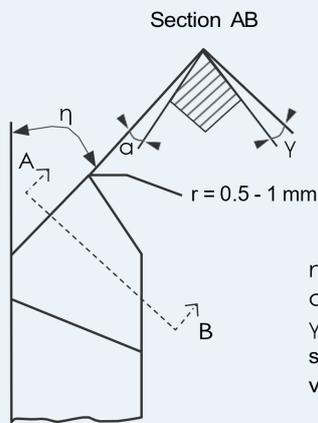
Turning & Boring

DIFFICULTY	MELTED SURFACE	ROUGH FINISH	BURRS AT EDGE OF CUT	CRACKING OF CHIPPING OF CORNERS	CHATTER
Common cause	<ul style="list-style-type: none"> • Tool dull or heel rubbing • Insufficient side clearance • Feed rate too slow • Spindle speed too fast 	<ul style="list-style-type: none"> • Feed too heavy • Incorrect clearance angles • Sharp point on tool (slight nose radius required) • Tool not mounted on center 	<ul style="list-style-type: none"> • No chamfer provided at sharp corners • Dull tool • Insufficient side clearance • Lead angle not provided on tool (tool should ease out of cut gradually, not suddenly) 	<ul style="list-style-type: none"> • Too much positive rake on tool • Tool not eased into cut (tool suddenly hits work) • Dull tool • Tool mounted below center 	<ul style="list-style-type: none"> • Too much nose radius on tool • Tool not mounted solidly • Material not supported properly • Depth of cut too heavy

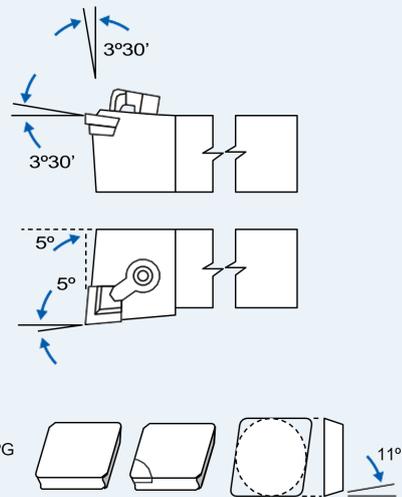
Parting (Cutting-off)

DIFFICULTY	MELTED SURFACE	ROUGH FINISH	SPIRAL MARKS	CONCAVE OR CONVEX SURFACES	NIBS OR BURRS AT CUT-OFF POINT	BURNS ON OUT-SIDE DIAMETER
Common cause	<ul style="list-style-type: none"> • Dull tool • Insufficient side clearance • Insufficient coolant supply 	<ul style="list-style-type: none"> • Feed too high • Tool improperly sharpened 	<ul style="list-style-type: none"> • Tool rubs during its retreat • Burr on point of tool 	<ul style="list-style-type: none"> • Point angle not great enough • Tool not perpendicular to spindle • Tool deflecting • Feed tool heavy • Tool mounted above or below center 	<ul style="list-style-type: none"> • Point angle not great enough • Dull tool • Feed too heavy 	<ul style="list-style-type: none"> • No chamfer applied before cut-off • Dull tool

DATA TURNING



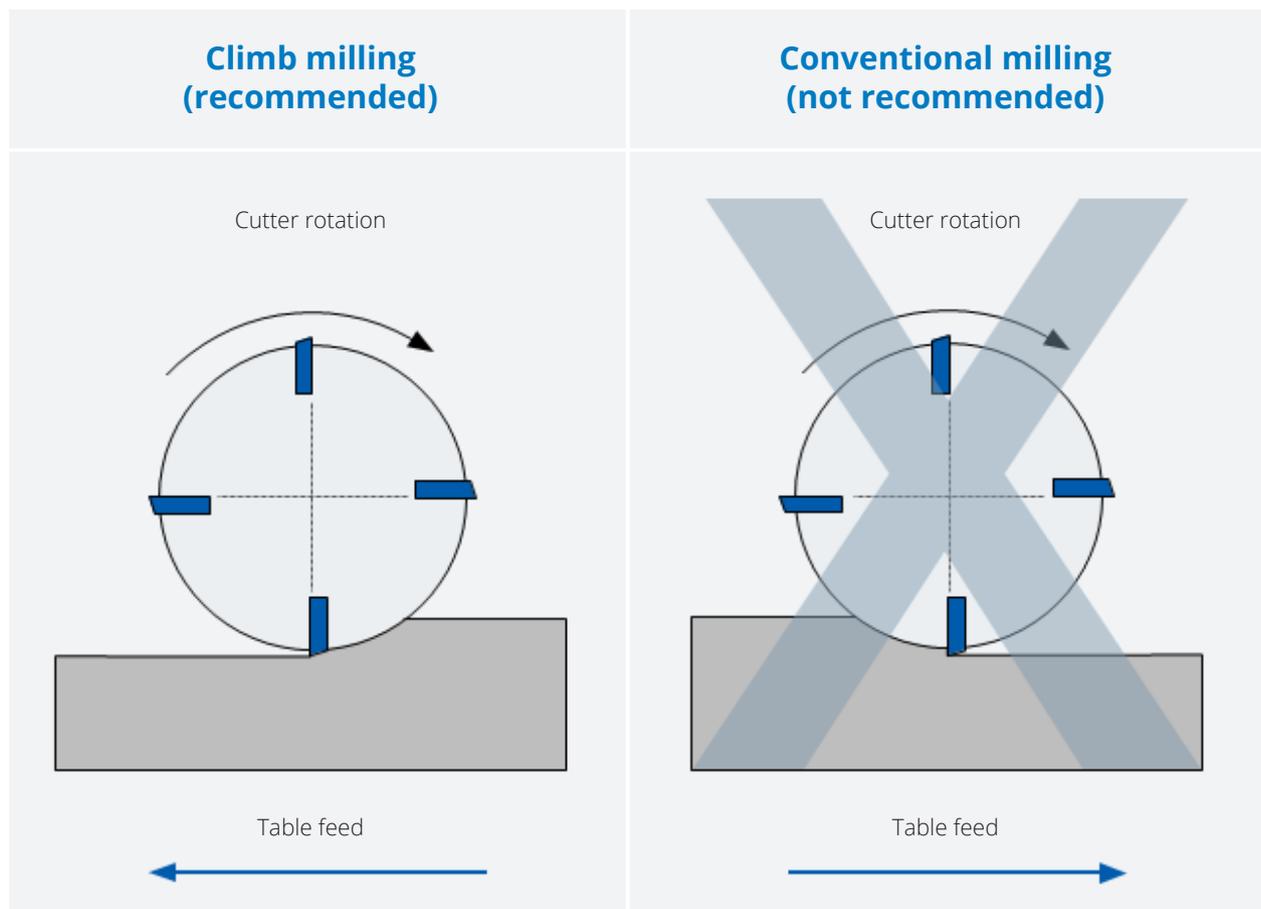
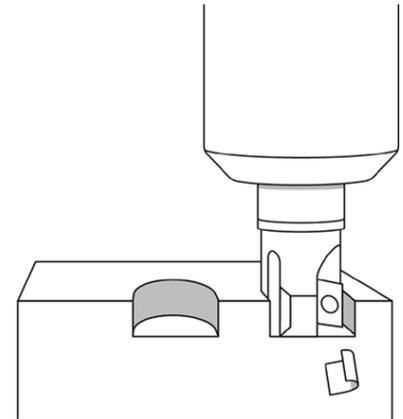
η : slide cutting edge angle (°)
 α : slide relief angle (°)
 γ : rake angle (°)
 s : feed (mm/rev.)
 v : cutting speed (m/min)



	α	γ	η	s	v	MATERIAL TOOL
Sustamid PA / Sustaglide PA, Polystone PE	5 - 15	0 - 10	0 - 45	0.05 - 0.05	200 - 500	HSS steel or carbide
Sustarin POM	5 - 15	0 - 10	0 - 45	0.05 - 0.05	200 - 500	HSS steel or carbide
Sustadur PET, PAI, SustaPEEK	5 - 15	0 - 10	0 - 45	0.05 - 0.05	200 - 500	Carbide
Palsun PC, Sustapei, Sustason PSU	5 - 15	0 - 10	0 - 45	0.05 - 0.04	200 - 400	Carbide
PAI, Sustapeek GF 30 / MOD / CF 30, Sustatron PPS MOD,	5 - 15	0 - 10	0 - 45	0.05 - 0.03	100 - 200	Carbide or PCD/CVD
PBI, PI	5 - 15	3 - 5	0 - 45	0.05 - 0.02	25 - 100	Carbide or PCD/CVD
Tetron/Ticomp/Tetco PTFE	8 - 12	0 - 5	0 - 45	0.05 - 0.04	150 - 400	Carbide

MILLING

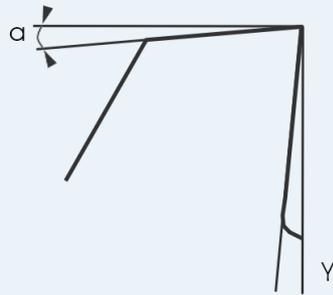
- Machines should have cavernous area for swarf.
- Use milling cutters with less teeth, high Avoid conveyor belts.
- For milling machines consider vacuuming dry chips.
- Use of Vacuum plates recommended for plate parts.
- Parts can be further supported with surrounds on vacuum plate.
- Avoid machining too much on one side of flat plates.
- Even stress out where pockets are required on one side with opposing face grooves.
- Consider using round billets in some cases to overcome small volume restrictions.



Small break, less heat, better finish

Large break, more heat, worse finish

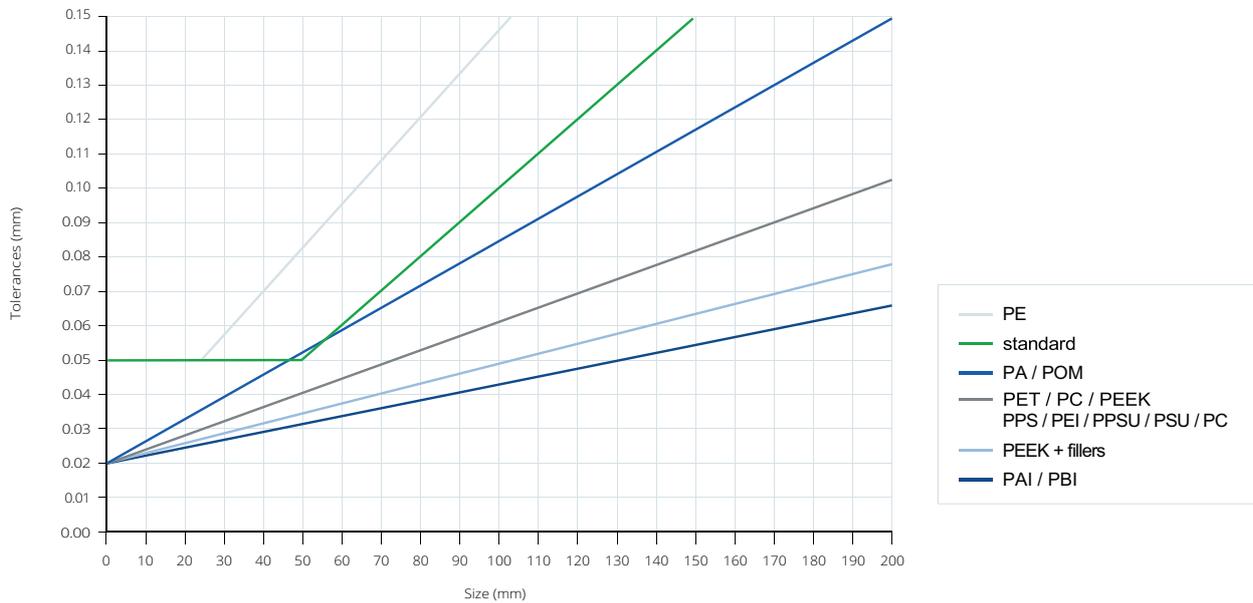
DATA MILLING



α: relief angle (°)
 γ: rake angle (°)
 s: feed (mm/tooth)
 v: cutting speed (m/min)

	α	γ	η	s	v	MATERIAL TOOL
Sustamid PA / Sustaglide PA, Polystone PE	5 - 15	0 - 15	0 - 45	≤ 0.5	200 - 500	HSS steel or carbide
Sustarin POM	5 - 15	0 - 15	0 - 45	≤ 0.5	200 - 400	HSS steel or carbide
Sustadur PET, PAI, SustaPEEK	5 - 15	0 - 15	0 - 45	≤ 0.4	150 - 300	Carbide
Palsun PC, SustaPEI, Sustason PSU	5 - 15	0 - 15	0 - 45	≤ 0.4	200 - 400	Carbide
PAI, SustaPEEK GF 30 / MOD / CF 30, Sustatron PPS MOD,	5 - 15	0 - 15	0 - 45	≤ 0.3	75 - 150	Carbide or PCD/CVD
PBI, PI	5 - 15	0 - 15	0 - 45	≤ 0.15	25 - 100	Carbide or PCD/CVD
Tetron/Ticomp/Tetco PTFE	8 - 12	0 - 15	0 - 45	≤ 0.3	100 - 250	Carbide

BEST TOLERANCES AT MCG ENGINEERED SOLUTIONS



These extra-precise tolerances can require special tooling and challenging machining steps which may lead to extra costs.

SIZE	PE	PET/ PC/ PEEK/ PPS/ PEI / PPSU/ PSU/ PC	PA/ POM/PTFE	PEEK WITH FILLERS	PAI / PBI
1	0.05	0.02	0.02	0.02	0.02
5	0.05	0.02	0.02	0.02	0.02
10	0.05	0.02	0.03	0.02	0.02
15	0.05	0.03	0.03	0.02	0.02
20	0.05	0.03	0.03	0.03	0.02
25	0.05	0.03	0.04	0.03	0.03
30	0.06	0.03	0.04	0.03	0.03
35	0.06	0.03	0.04	0.03	0.03
40	0.07	0.04	0.05	0.03	0.03
45	0.08	0.04	0.05	0.03	0.03
50	0.08	0.04	0.05	0.03	0.03
60	0.10	0.04	0.06	0.04	0.03
70	0.11	0.05	0.07	0.04	0.04
80	0.12	0.05	0.07	0.04	0.04
90	0.13	0.06	0.08	0.05	0.04
100	0.15	0.06	0.09	0.05	0.04
110	0.16	0.07	0.09	0.05	0.05
120	0.17	0.07	0.10	0.05	0.05
130	0.18	0.07	0.10	0.06	0.05
140	0.20	0.08	0.11	0.06	0.05
150	0.21	0.08	0.12	0.06	0.05
160	0.22	0.09	0.12	0.07	0.06
170	0.23	0.09	0.13	0.07	0.06
180	0.25	0.09	0.14	0.07	0.06
190	0.26	0.10	0.14	0.08	0.06
200	0.26	0.10	0.15	0.08	0.07

THREAD INSERTS

Threaded inserts are not required when machining most of our engineering materials. They are, however, recommended where there are strong vibrations or repeated clamping and unclamping.

Self-tapping Tappex is most popular, however, be cautious!

- If using slotted inserts pre-tab a part of the thread.
- Check that there is no closure after insertion.
- Consider inserts with cross hole to cut.



Some can be inserted by heat. This is slower but fine for General Engineering Plastics.

- Use of spiral taps is recommended (not straight).
- Where possible use rigid tap CNC cycles.
- Check pilot drill size.
- Go for high quality taps where possible.
- Turning threads should be straight forward.
- Tools do not have to be as rigid as metal cutting tools.

DEBURRING

- Sharp knives or rotating hooks are most commonly used. Tool sharpness is crucial.
- Counter sink tools like this are recommended as they do not cut too deep or chatter.
- Hard plastics can slightly be abraded by applying wet and dry process.
- Rotary stones can be effective on some plastics.
- Hot air is good for complex profiles made from General Engineering Plastics.



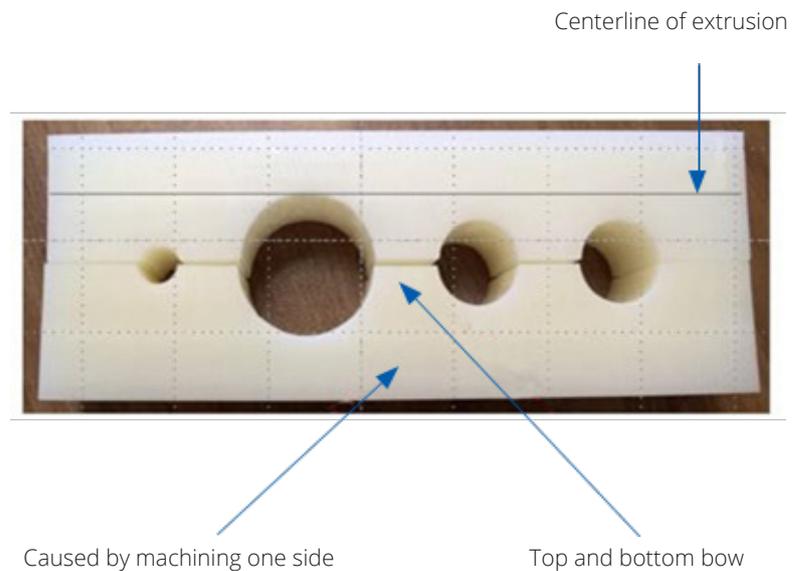
GENERAL RULES

- Consider rod as well as plate.
- Machine equally to the centerline.
- If you have to machine pockets perform intermediate steps.



EXAMPLE

1. Use cast polyamide plate.
2. Cut out the section so that it is symmetrical to the center line of the extrusion (90° to the cut-out).
3. Start with thickest plate and move extrusion line away from machined features.
4. Machine slots/grooves in opposite face.
5. Balanced machining on both sides of the shape's centerline can also help prevent warpage.



ANNEALING

Better flatness and tighter tolerances

Extremely close-tolerance parts requiring precision flatness and non-symmetrical contour sometimes require intermediate annealing between machining steps. Improved flatness can be attained by rough machining, annealing and finish machining with a very light cut.

Recommended annealing procedure

T1: Heat-up time (heating rate: 10-20 °C/h)

T2: Hold-time

depends on the wall thickness: 10 min/mm T3: Cool-down time (cooling rate: 5-10 °C/h)

T4: Additional time required to establish room temperature depends on the wall thickness: 3 min/mm

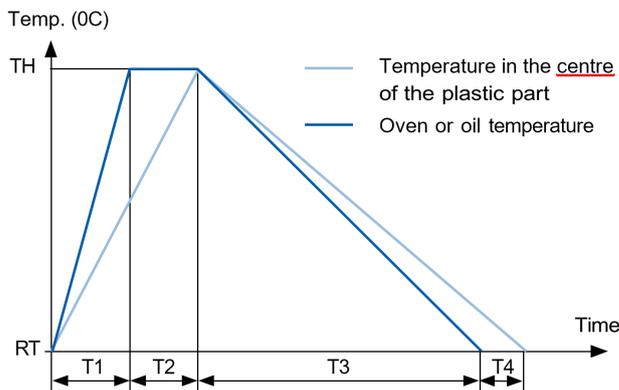
Annealing Tips

When pre-machining, leave enough oversize to allow machining to final sizes after annealing.

Fixturing parts to desired shape or flatness during the entire annealing cycle often proves advantageous.

Do not unfix until parts have completed entire annealing cycle and cooled down slowly to the touch.

Make sure that temperatures are uniform and within $\pm 3^{\circ}\text{C}$ all over the oven or the oil bath at all times during annealing process.



RT: room temperature TH: hold-temperature

MATERIALS	HEAT-UP	HOLD TEMPERATURE	HOLD-TIME (T2)	COOL DOWN	ENVIRONMENT*
PA	10 – 20°C / h	150°C	10 min / mm	5 – 10°C / h	air, nitrogen or oil
POM	10 – 20°C / h	150°C	10 min / mm	5 – 10°C / h	air, nitrogen or oil
PET	10 – 20°C / h	150°C	10 min / mm	5 – 10°C / h	air, nitrogen or oil
PE	10 – 20°C / h	120°C	10 min / mm	5 – 10°C / h	air, nitrogen or oil
PC	10 – 20°C / h	130°C	10 min / mm	5 – 10°C / h	air or nitrogen
PEEK	10 – 20°C / h	250°C	10 min / mm	5 – 10°C / h	air or nitrogen
PPS	10 – 20°C / h	200°C	10 min / mm	5 – 10°C / h	air or nitrogen
PPSU	10 – 20°C / h	200°C	10 min / mm	5 – 10°C / h	air or nitrogen
PEI	10 – 20°C / h	200°C	10 min / mm	5 – 10°C / h	air or nitrogen
PSU	10 – 20°C / h	170°C	10 min / mm	5 – 10°C / h	air or nitrogen

SUMMARY

When machining stock shapes remember...

- Thermal expansion rate is up to 10x higher 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.

Because of these significant differences you may wish to experiment with fixtures, tool materials, angles, speeds and feed rates to obtain optimum results.

All statements, technical information, recommendations, and advice are for informational purposes only and are not intended and should not be construed as a warranty of any type or term of sale. The reader, however, is cautioned that Dotmar Engineering Plastics does not guarantee the accuracy or completeness of this information and it is the customer's responsibility to test and assess the suitability of the products in any given application or for use in a finished device.



Australia

1800 170 001 | dotmar.com.au

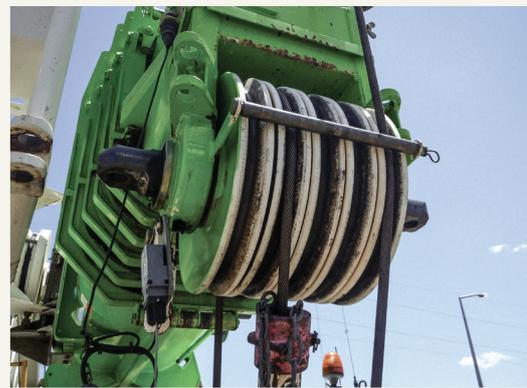
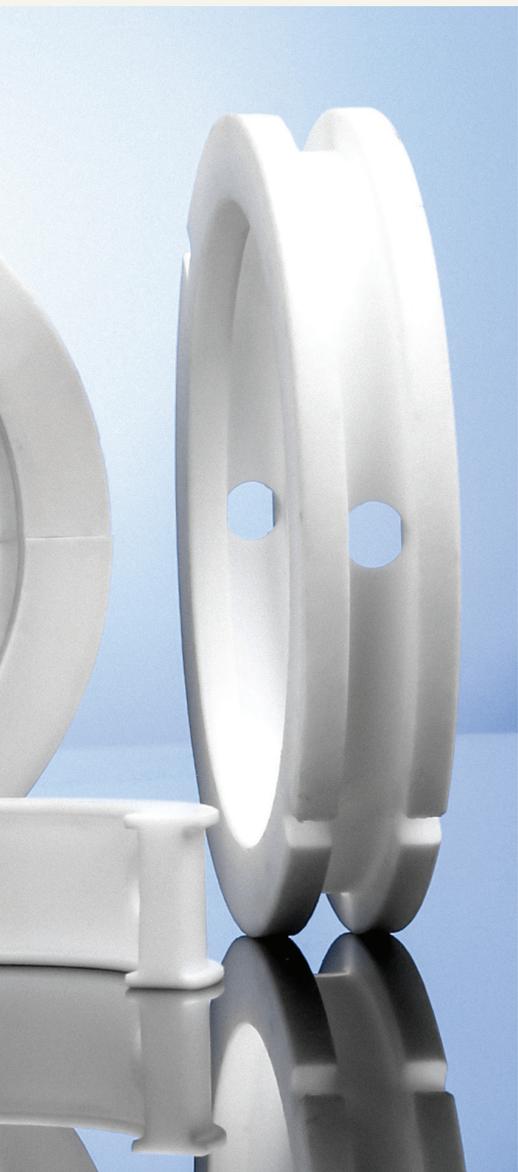
VICTORIA | NEW SOUTH WALES | QUEENSLAND
SOUTH AUSTRALIA | WESTERN AUSTRALIA

New Zealand

0800 650 750 | dotmar.co.nz

AUCKLAND | CHRISTCHURCH | PALMERSTON NORTH





DOTMAR

SINCE 1967