SKYGREEN

Glycol Modified Copolyester Resin

Mould Design & Processing Guidelines for Injection Moulding

CONTENTS

INTRODUCTION TO SKYGREEN® COPOLYESTERS

SKYGREEN® Copolyesters are Glycol Modified Copolyester resins manufactured by SK Chemicals for injection moulding applications. SKYGREEN® copolyesters offers excellent clarity, toughness, and processability.
This document contains processing recommendations for the SKYGREEN® product range. The recommendations are given in good faith, but due to wide variations in both processing equipment and product design it is not possible here to provide accurate information for any particular situation. To discuss any particular manufacturing process the customer is invited to contact the SK Chemicals Technical Service Team, details of which are contained at the end of this document.

What is SKYGREEN®? What is Glycol Modified Copolyster Resin?
SKYGREEN® is an amorphous copolyester produced by the reaction of terephthalic acid (TPA) with ethylene glycol (EG) in which a certain amount of the ethylene glycol is replaced with cyclohexane dimethanol (CHDM) – see Fig 1. The addition of CHDM prevents crystallisation, leading to improved processability combined with outstanding toughness, clarity and chemical resistance.

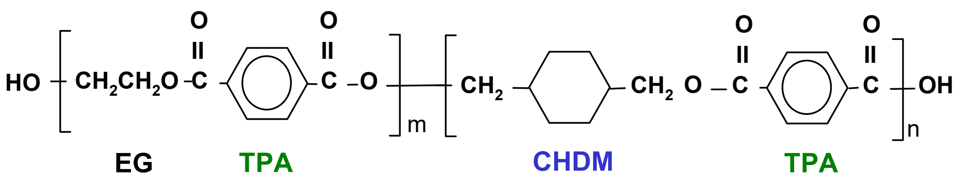


Fig. 1 – Molecular structure of SKYGREEN® copolyester

What is CHDM?
CHDM (1,4-Cyclohexane dimethanol) is derived from DMT (Dimethyl terephthalate) by hydrogenation. The main applications include the manufacture of chemically modified PET, amorphous PET (PETG & PCTG), engineering plastics and adhesives.

SK chemicals

SKYGREEN

DATA SHEET

Preliminary Data Sheet

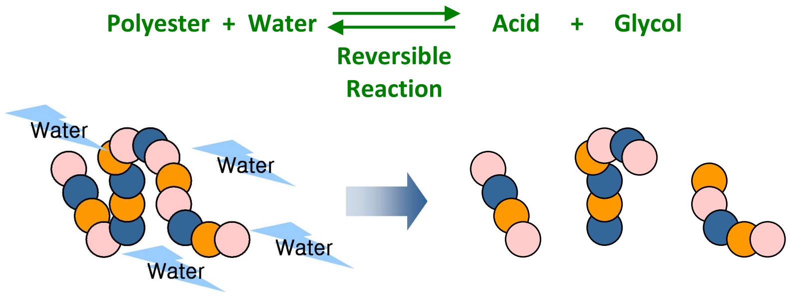
Injection Moulded Properties (ASTM Method)

Properties	Test Method	Unit	Typical Values KN100/KN200	JN100/JN200	PN
Physical					
Specific Gravity	ASTM D792	-	1.27	1.23	1.27
Rockwell Hardness	ASTM D785	R scale	108	105	105
Water Absorption (24 hr immersion)	ASTM D570	%	0.13	0.13	0.13
Mechanical					
Tensile Strength @ Yield 50mm/min (2 inch/min)	ASTM D638	MPa (kgf/cm^2)	50 (510)	45 (460)	50 (510)
Tensile Strength @ Break 50 mm/min (2 inch/min)	ASTM D638	MPa (kgf/cm^2)	29 (300)	53 (540)	26 (265)
Elongation @ Yield 50 mm/min (2 inch/min)	ASTM D638	%	4.5	5.0	4.5
Elongation @ Break 50 mm/min (2 inch/min)	ASTM D638	%	180	340	120
Flexural Strength 1.27mm/min (0.05 inch/min)	ASTM D790	MPa (kgf/cm^2)	69 (700)	67 (685)	71 (720)
Flexural Modulus 1.27mm/min (0.05 inch/min)	ASTM D790	MPa (kgf/cm^2)	2100 (21400)	1800 (18300)	2150 (21900)
Izod Impact Strength Notched @ 23°C (73°F)	ASTM D256	J/m (kgf.cm/cm)	105 (10.7)	NB	85 (8.7)
Thermal					
Heat Distortion Temperature @ 0.455 MPa (66 psi) @ 1.82 MPa (264 psi)	ASTM D648	°C (°F)	70 (158) 62 (143)	74 (165) 64 (147)	70 (158) 62 (143)
Optical					
Haze	ASTM D1003	%	< 1.0	< 1.0	< 1.0
Total Transmittance	ASTM D1003	%	90	91	91
Flammability					
UL Flammability Classification @ 3.0mm thickness	UL 94	-	HB	HB	-

The data listed here consists of preliminary data for this range of products. Therefore this data should not be used to establish specification limits or be used alone as a basis for design. This information is not intended as a warranty of any kind. Customers must make their own representative tests and assume all risks of use, whether used alone or in combination with other products. SK Chemicals assumes no obligation or liability for any advice furnished or results obtained with respect to these products. All warranties of merchantability for particular purpose or use are excluded and disclaimed.

DRYING

SKYGREEN® copolyester resin has a tendency to absorb atmospheric moisture, which can cause hydrolytic degradation during processing. This results in a decrease in molecular weight of the resin and in a reduction of the physical properties of the final product.



In order to avoid this degradation, SKYGREEN® copolyester resin should be sufficiently dried to a moisture level of less than 600 parts per million (ppm) before processing. The drying operation is most efficiently performed by using a commercially available recirculating, dehumidified hot air drying system similar to that shown in Fig 3. Further details of suppliers of suitable drying equipment can be obtained from the SKYGREEN® Technical Service Team.

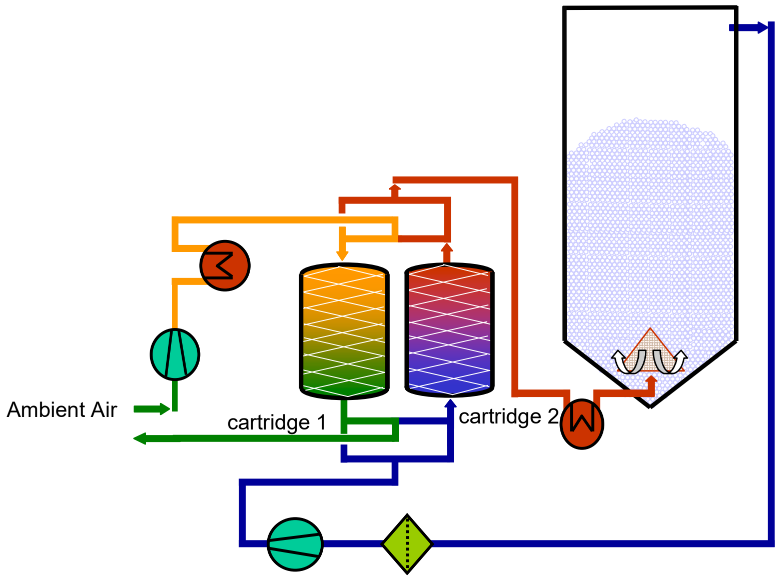


Fig. 3 – Typical drying system

SKYGREEN® copolyester resin is loaded into the drying hopper where heated, dehumidified air is passed through the material. The moisture extracted from the granules in the drying hopper is absorbed by the desiccant cartridge 2. At the same time, the desiccant in cartridge 1 is regenerated. Once the desiccant in cartridge 2 becomes saturated the drying operation is changed to cartridge 1 whilst cartridge 2 is then regenerated.

The optimum drying conditions for SKYGREEN® are as follows:

Drying Temperature	65°C (165°F) - 70°C for JN100 and JN200
Drying Time	4 to 8 hours
Air Flow of Dry Air	> 0.065 m3/min per kg/h (1cfm per lb/h)
Dew Point of Dry Air	< -30°C (-20°F), preferably -40°C (-40°F)
Residual Moisture Content	<0.06% (600 ppm)
Drying Hopper Capacity	6 to 8 times extruder output per hour
Height/Diameter Ratio	> 2:1, 3:1 is better for plug flow
Insulation	Insulate well to improve energy efficiency
Monitoring	Inlet Air Temperature & Dew Point

Insufficient predrying is the most common cause of processing problems such as:

- Molecular weight (I.V) reduction of the polymer and degradation of any additives.
- Adverse effect on the colour of the final product.
- Difficult control of processing parameters such as melt pressure, rheology, and power consumption.
- Bubbles and silver streaks.

MOULD DESIGN

Cooling: Mould cooling is crucial to prevent sticking, to aid part ejection, and to reduce cycle times. Cooling is especially critical in cores, corners, and tall-standing details and near injection gates. If possible, use thermally efficient Beryllium copper type materials for cores and other hard-to-cool areas of the mould. Design cooling circuits for adequate turbulent flow and minimise cooling line loops.

Cold Sprue Design and Cooling: Copolyester polymers can tend to stick to hot steel. To prevent sprue sticking:

- Use upper and lower cooling circuits around the sprue.
- Use a slight interference fit between the sprue bush and mould plate to ensure good heat transfer.
- Use thermally efficient alloy bushings.
- Keep sprues shorter than 80 mm with an included draft angle of 2 ° minimum.
- Polish the sprue in the ejection direction.

The following factors must be taken into consideration when determining the sprue design :

- Length and cross-sectional area of the runners
- The size of moulded product and the injection machine used

The standard shape of sprue is shown in Fig. 1.

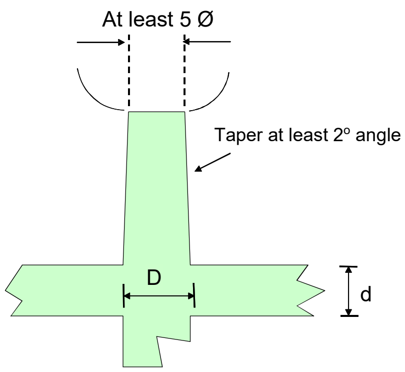


Fig. 1. Standard Sprue design

Cold Runner Design: Use the same guidelines that apply to most engineering resins when designing cold runner systems. Including designs for smooth and balanced flow, generously radiused transitions, cold slug wells, runner venting, and full- or half-round cross sections.

The SKYGREEN® copolyester resin is injected into the cold manifold of the runners at high speed. Cooling takes place on the channel walls. The cross-sectional areas of the runner must therefore be large enough to ensure the follow-up flow of the pressurised material. This follow-up flow of material prevents the volumetric contraction of the moulding.

The runner should be of such a design as to achieve favourable flow cross-sectional areas. This will always be the case with round channels having been distributed as equal half sections in both mould halves. (Fig. 2)

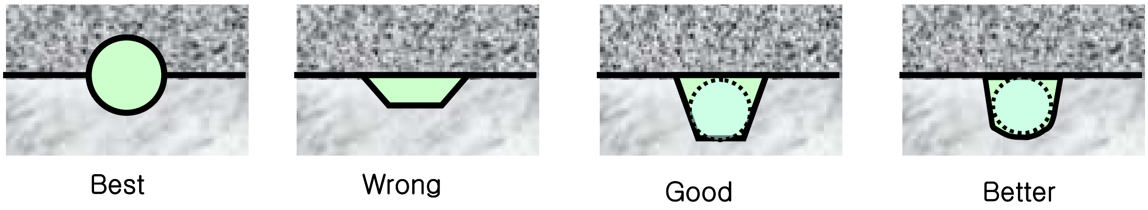


Fig. 2. The most efficient runner design

In case of round runner, this increases the cost of the mould of course, so half-round runner may be good.
To eject well from the mould, a 4~5 º draft angle on the flat sides of the runner is recommended.

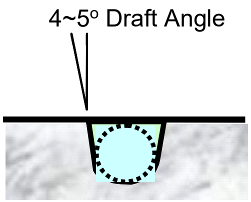
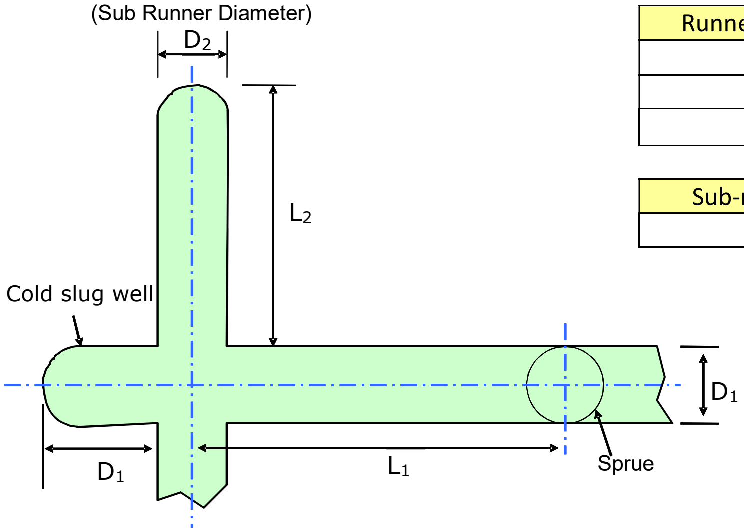


Fig. 3. Recommended draft angle of runner

Secondary runners branch off from the main runner. These secondary runners should branch in radiused bends and not at right angles.

The runner and sub-runner should be designed in diameter and length, as recommended below. Also, a cold slug well should be provided at the bent section.



Runner length, L1 (mm)	Runner Diameter, D1 (mm)
< 70	6
70 ~ 200	8
> 200	More than 10

Sub-runner length, L2	Runner Diameter, D2
< 70	6

Fig. 5. General guideline of runner dimension

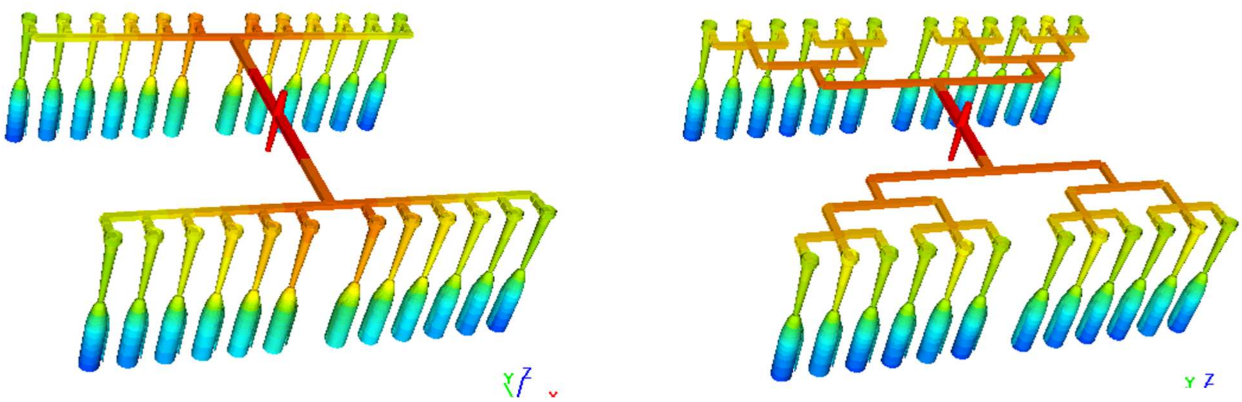


Fig. 6. Poor balanced runner system vs Well balanced runner system

Gate Design: In order to minimise gate blush and prevent premature gate freezing gates should be large enough and with smooth transitions. Edge and tab type gates should be 50% to 80% of the nominal wall thickness. Pin-gates in three-plate moulds are typically 0.75 to 1.1 mm in diameter.

The gate is given a smaller cross-section than the runner so that the moulding can be easily degated (separated from the runners). The position and dimensions of the gate are critical, and sometimes the gate must be modified after initial trials with the mould after considering how the resin fills, how the moulded products can be easily detached and how finishing can be facilitated without difficulty.
The following are examples of typical gate shapes and designs used for SKYGREEN® copolyester resin.

1) Fan gate
Provides a good spread of material over a large area.

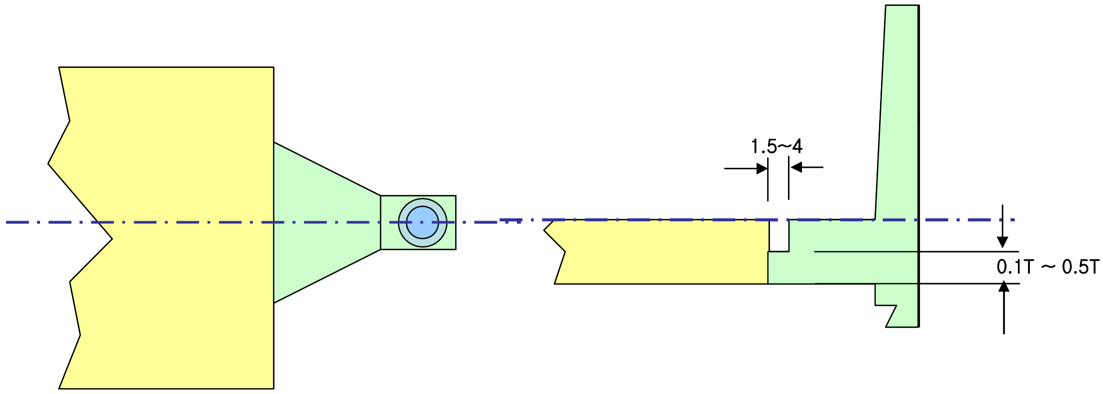
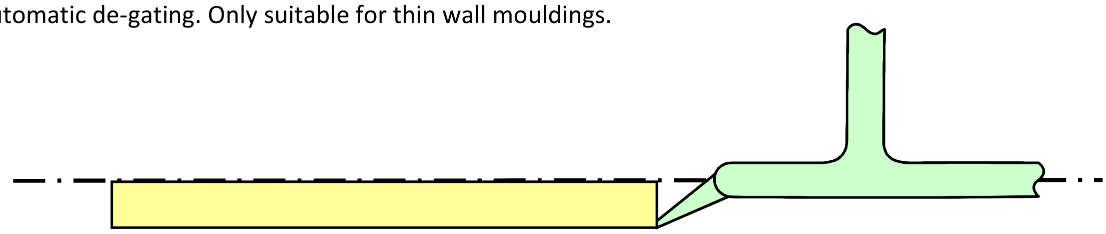
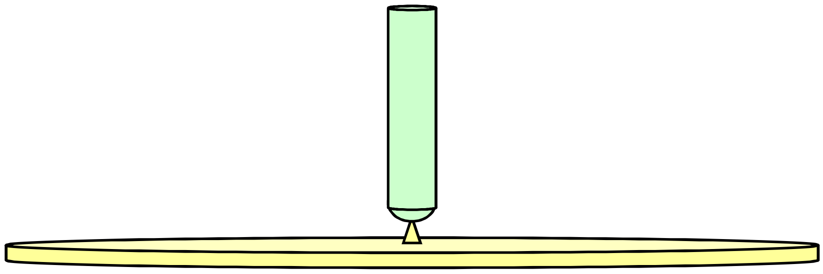


Fig. 7. General guideline for fan gate

2) Tunnel or submarine gate
Provides automatic de-gating. Only suitable for thin wall mouldings.



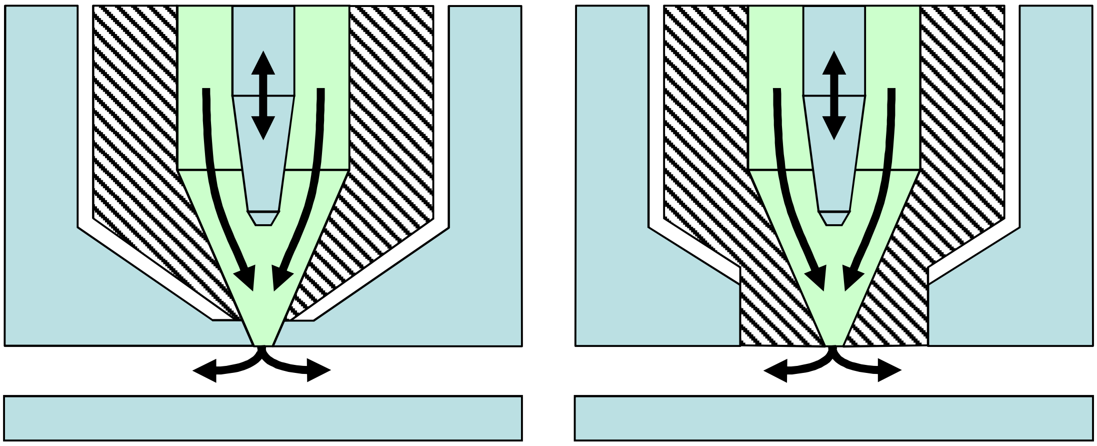
3) Pin gate
Used to minimise finishing where edge gating is undesirable and for automatic de-gating. Only suitable for thin sections



Hot Runners/Valve Gates: SKYGREEN® copolyesters work well in hot runner and/or valve-gated moulds that are properly designed for amorphous polymers. Many suppliers have designs specifically for copolyesters. Typically, these have external heating of a streamlined melt stream.

Because of the increased chance of degradation, types not recommended are those that have heaters inside the melt stream but no external heating or insulation.

Valve gates provides the cleanest gate combined with less restriction of the injected material. Large gate diameter permits holding pressure to be maintained to reduce sink marks caused by thermal contraction of the material.



Recommended design: cavity wall temperature near to the injection point can be maintained at lower temperature to prevent sticking.

Not recommended: The high temperature sprue bush can cause sticking problems near to the injection area.

PRODUCT DESIGN

Venting: For vents close to gates or for small parts the typical vent depth is 0.012 to 0.025 mm. Up to 0.035 mm may be needed for larger parts. The land is typically 3 to 6 mm. Polished vents are preferable if a mould release additive is used.

Mould Surface Polish/Coatings: Mould surfaces should be polished to aid ejection. Mirror polish if required for aesthetics. However, a surface that is too smooth can create a vacuum during ejection. Surface coatings may be used where cooling is not adequate to prevent sticking.

Draft Angles and Textured Surfaces: In polished moulds, 1° draft per side is suggested to aid ejection. However, 0.5° draft can be used to obtain reasonable dimensions in ribs, bosses, or other design features. Avoid using zero draft in any part of the mould. Texturing requires increasing the draft angle by 1 ° to 1.5 ° for every 0.025 mm of texture depth.

Ribs and Radii: Ribs and walls should meet with a minimum radius of 0.5 mm or 0.2 to 0.5 times the wall thickness, whichever is greater. This will minimise stress concentration in moulded parts and disturbance of flow during filling.

Undercuts: Stripped undercuts such as snap rings, or threads are allowable up to 2% to 3% of the part diameter in relatively thin-walled parts. Undercuts must be rounded and well filleted to allow proper ejection.

PROCESSING

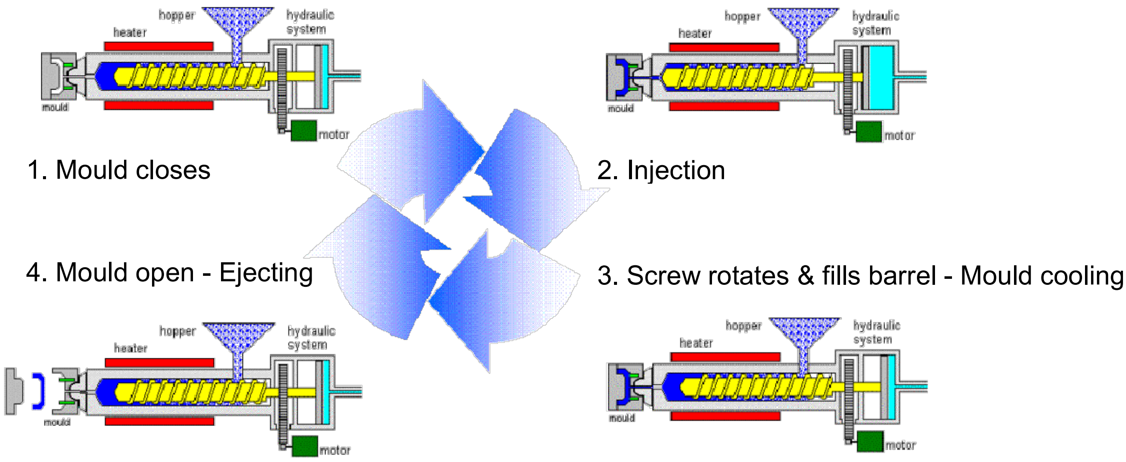
SKYGREEN® copolyester resin can be successfully processed using conventional injection moulding equipment. The following information contains some general processing guidelines for SKYGREEN® copolyester resin in order to obtain the best quality product.

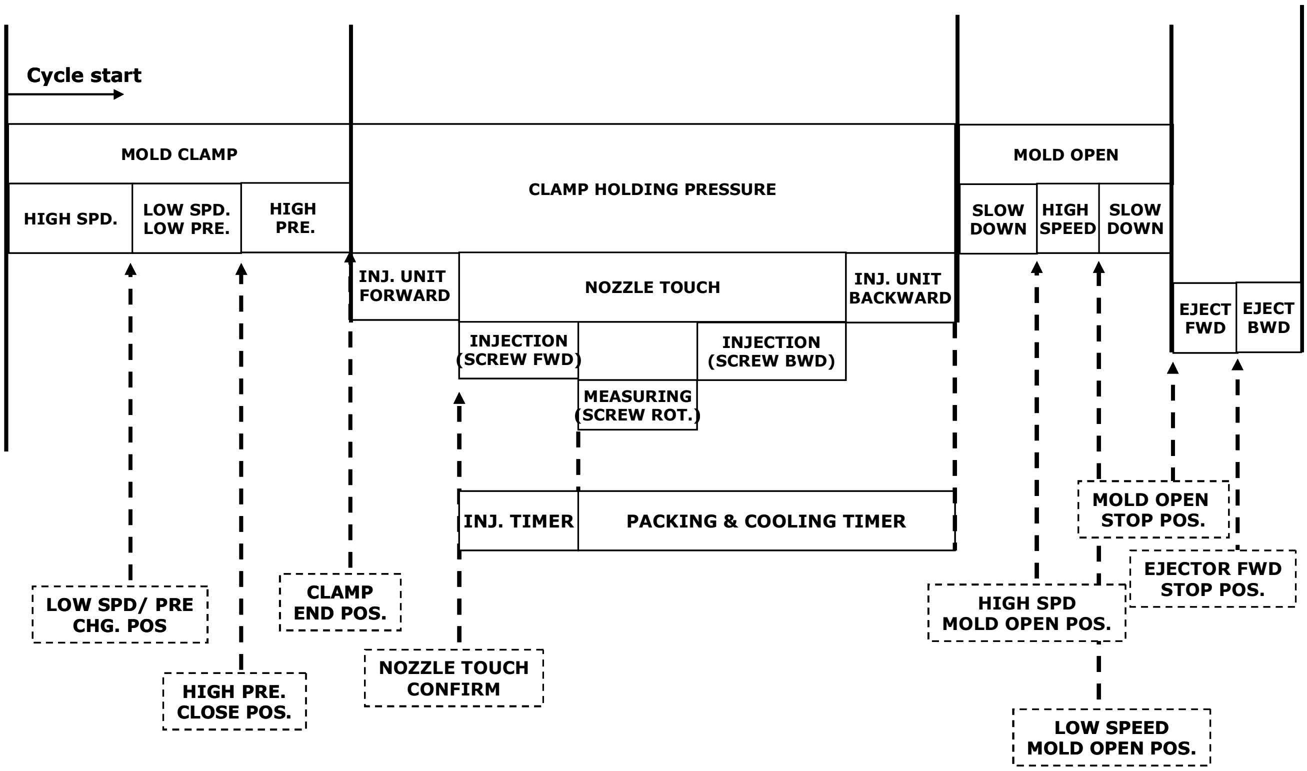
INJECTION MOULDING

Moulding Cycle

Following figure is typical moulding cycle of thermoplastics for injection moulding.

Fig. 4 – Typical Injection Moulding Cycle





MOULDING MACHINE

Shot Capacity/Clamping Tonnage: Actual shot size for moulded parts, including runners, should be 40% to 70% of shot capacity to provide adequate plasticising time without excessive hold-up time. Clamping force is typically 400 to 700 kg/cm^2 (3 to 5 tons/in.^{2}) of projected area of the part and runner.

Injection Profiling: The moulding machine should be capable of profiling injection speed. Profiling is generally necessary to optimise the filling stage of the injection moulding process.

Screw and Barrel Design: SKYGREEN® copolyesters generally cause little wear to the screw and barrel, and no corrosion is expected. General-purpose screws with compression ratios of 2.8:1 to 3.2:1 and length-to-diameter ratios of 18:1 to 20:1 have been used successfully. Vented barrels are not recommended. Ring-type non-return valves are preferred over ball-type valves to minimise residence time and excessive degradation.

Melt and Mould Temperatures: Actual melt temperatures for SKYGREEN® copolyesters typically range from 250 ºC to 280 ºC. Use the minimum temperature necessary to fill the part without excessive speed or pressure. When restarting the machine, or on initial start-up, extrusion should be begun as soon as possible after the extruder reaches the set temperatures.

Typical mould temperatures for SKYGREEN® copolyesters range from 15 ºC to 25 ºC depending on aesthetic and filling requirements.

Injection Speed: Generally, use slower initial injection speed than for other plastics to minimize gate blush and/or splay. Inject at 10% to 20% of available speed for the first 10% of cavity fill, then increase to 40% to 80% speed to complete the shot.

Cushion: Keep the cushion size at a minimum to prevent degradation from excessive residence time. Typical cushion is 3 to 13mm.

Back Pressure and Decompression: Typical back pressure is 0.7 to 1.0 MPa, although higher pressures up to 2 MPa may be needed to eliminate air entrapment and improve melt uniformity. In general, little or no decompression is used.

Production interruptions: If production is stopped for more than about 15 to 20 minutes, the extruder temperatures should be reduced to about 150ºC in order to reduce excessive degradation of the polymer.

Purging and Shutdown: On machine shut-down it is recommended that purging materials such as PP or PE should not be used. Just increase the extruder temperatures by 20ºC and drain as much material as possible. Always leave the screw forward after shutdown to ease restarting.

Regrind: Clean, dried regrind can be used at blend rates up to 100%, although 10% to 30% is more typical. Keep blend rates uniform to minimise process variation. Since regrind has somewhat lower physical properties than virgin polymer, be sure that parts moulded with regrind meet the end-use requirements.

SKYGREEN® offers the high melt-flow and non-crystalline properties required for successful injection moulding. The following processing data are based on those used to produce injection moulded components in the SKYGREEN® R&D Laboratory:

Screw type	General purpose with L/D = 18:1 to 20:1 and compression ratio: 2.5:1 to 3.5:1 Ring-type non-return valve preferred.					
Temperature (°C)	Mould	Nozzle	C2	C3	C4	Feeding
	40	245	245	245	210	240
	** For J series, temperatures can be increased by 5 to10 °C.*					
Pressure (bar)	1st Pressure: 55 2nd Pressure: 40, 40, 35 Back Pressure: 5	Speed (mm/sec)	V12: 8 V21: 8			
Time (sec)	Injection: 2.0 Hold: 15 Cooling: 15 Cycle: 36	Screw	Refill speed: 50% Rpm: 100 Stroke: 62 mm			
General guidelines:	• Injection speed: slow to medium. Screw speed: 50 to 100 rpm. • Cushion size: minimum (3 to 13mm). Decompression: To minimize drooling. • Back pressure: 0.3 to 1MPa (50 to 100psi) is enough to give uniform metering & reduce air entrapment. • Hold pressure / time: To eliminate sink marks or voids, but avoid overpacking.					

TROUBLESHOOTING GUIDE

BLACK SPECKS

Possible Solutions...

- Be sure runner system is streamlined.
- Check the mould for blocked / adequate venting.
- Check for contaminated material, particularly if regrind is being used.
- Ensure both screw and barrel have been purged clean.
- Check for mismatched nozzle.
- Decrease injection pressure.
- Decrease melt temperature to minimise degradation.
- For long cycle times, decrease rear barrel temperatures.
- Examine heater bands and controls for malfunctions. Locate any hot spots.
- Examine nozzle seat, cylinder walls and valves for dead spots.
- Examine screw, screw tip and end cap for charred material or for rough surfaces.
- Increase melt temperature and purge to loosen degraded resin and to reduce trapped air.
- Minimise fines in raw material.
- Reduce back pressures.
- Reduce screw speed.
- Reduce injection speed.

BRITTLENESS

Possible Solutions...

- Adjust injection pressure.
- Be certain material has been dried to processing guidelines. Over dried material too could be a problem.
- Check for voids.
- Check and lower melt temperature.
- For long cycle times, decrease rear barrel temperatures.
- Check regrind levels.
- If gate is too thin, increase gate size.
- Improve cooling channels to prevent heat build up.
- Increase mould temperatures.
- Reduce back pressure.
- Moisture condition parts.
- Part designed with abrupt wall section transitions, sharp fillets, or radii.

BLISTERS

Possible Solutions...

- Increase injection back pressure.
- Lower injection speed.
- Material might be contaminated.
- Properly dry material to recommended guidelines.
- Raise mould temperature.
- Reduce screw rpm.
- Regrind not mixed sufficiently with Virgin material.
- Relocate gate.

BUBBLES

Possible Solutions...

- Be certain material is dry.
- Check for blocked vents.
- Change the location of vents.
- Increase feed zone temperature.
- Increase mould temperature.
- Increase hold / pack time.
- Increase injection pressure.
- Increase the number of vents.
- Match nozzle and sprue.
- Minimise or eliminate suck back.
- Increase/ maintain consistent cushion.

Note: Bubbles and Voids are two different problems.

BLUSH

Possible Solutions...

- Check and adjust melt temperature.
- Check and adjust nozzle temperature.
- Break edges around gate.
- Increase gate diameter.
- Increase injection pressure.
- Increase mould temperature.
- Increase the number and/or size of cold wells.

BURNS

Possible Solutions...

- Add radius to sharp corners.
- Check and adjust melt temperature.
- Check screw rpm and adjust accordingly.
- Check residence time of polymer in the barrel.
- Check for blocked vents.
- Check for correct vent dimensions.
- Check nozzle temperature.
- Check vent cavity at point of air entrapment.
- Decrease back pressure.
- Decrease injection pressure.
- Decrease injection speed.
- Examine heater bands and controls for malfunction - locate hot spots.
- Increase gate dimensions.
- Increase mould temperature.
- Increase nozzle diameter.
- Relocate gate.
- Use a lower compression ratio screw.
- Vent the runner system.

CRACKS

Possible Solutions...

- Increase mould temperature.
- Lower hold / pack pressure.
- Minimise sharp corners on part.
- Reduce fill rates.
- Slow ejector speed.
- Slow mould opening speed.

DELAMINATION / LAMINATION

Possible Solutions...

- Be certain material is dry.
- Check for contamination.
- Check for sufficient venting.
- Decrease residence time; Shorten overall cycle.
- Decrease screw speeds.
- Examine heater bands and controls for malfunction.
- Examine resin for contamination and foreign materials.
- Increase injection speed if inadequate.
- Increase / reduce melt temperature.
- Lower nozzle temperature.
- Reduce back pressure.
- Remove sharp corners at gate.
- Set mould temperature to recommended conditions.
- Use higher back pressure - change screw speed - use a higher compression screw.

DISCOLORATION

Possible Solutions...

- Be certain material is dry.
- Check material and containers for contamination.
- Check for sufficient venting, clean if needed.
- Decrease residence time, Shorten overall cycle.
- Decrease screw speeds.
- Ensure screw and barrel have been purged clean.
- Examine heater bands and controls for malfunctions.
- Reduce nozzle temperature.
- Reduce melt temperature.
- Reduce back pressure.

EXCESSIVE SHRINKAGE

Possible Solutions...

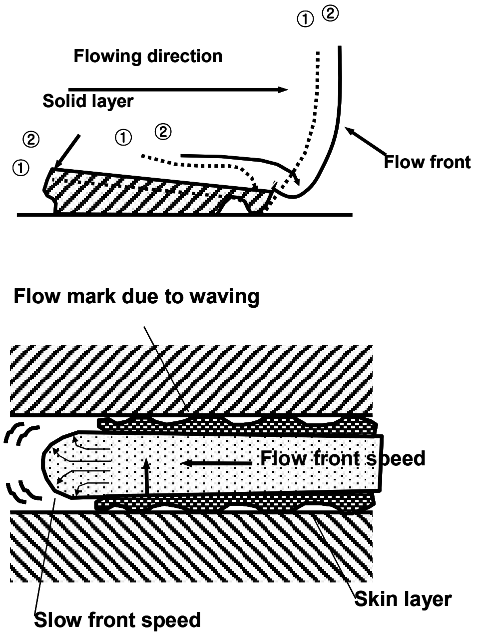
- Check gate location and relocate as needed.
- Check for a worn check valve assembly, barrel or screw.
- Decrease gate land length to improve cavity packing.
- Decrease mould temperature. Colourants may affect shrinkage.
- Ensure there is an adequate cushion.
- Increase injection hold time, and / or pressure.
- Increase injection pressure.
- Increase cooling time.
- Reduce melt temperature.

FLASH

Possible Solutions...

- Review projected area vs. clamp pressure.
- Profile the injection pressure: Low pressure at start, higher pressure to finish.
- Check for proper vent dimensions, clean if needed.
- Reduce melt temperature.
- Be certain material is dry.

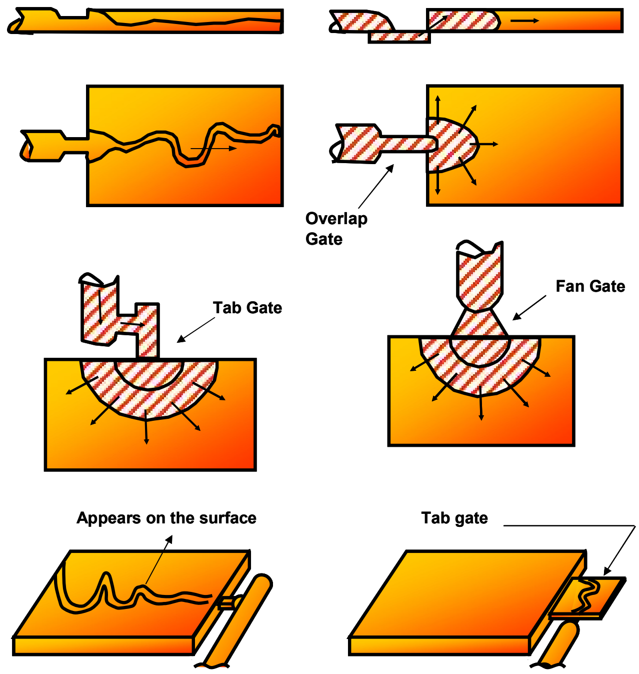
FLOW LINES



Possible Solutions...

- Add radius to sharp corners.
- Check for colour concentrate carrier compatibility.
- Decrease injection speed.
- Increase back pressure.
- Increase cold slug well size or number.
- Increase gate size and/or relocate as needed.
- Increase melt temperature.
- Increase mould temperature.
- Increase runner size.

JETTING



Possible Solutions...

- Check and adjust melt temperature.
- Change gate location so material will hit opposite wall.
- Check nozzle heater band.
- Decrease gate land length.
- Decrease injection speeds
- Flare the gate.
- Increase mould temperature.
- Increase sprue, runner or gate size.

SURFACE FINISH (LOW GLOSS)

Possible Solutions...

- Adjust injection pressure / speed.
- Be certain material is dry.
- Check for material contamination.
- Check for sufficient feed, increase cushion accordingly.
- Check that sprue, runner and gate are a sufficient size.
- Clean mould surface.
- Increase gate size.
- Increase melt temperature.
- Increase mould temperature.
- Prepare mould surface sufficiently to give the gloss expected for the moulded item.
- Provide adequate venting.
- Relocate gate so that melt flow hits the mould surface, not directly into cavity.
- Remove and clean mould.
- Vent cavity at the point of air entrapment.

MOULD FLASH

Possible Solutions...

- Adjust clamp pressure to maximum for that mould / press combination.
- Balance cavities to within 90% of fill.
- Be certain material is dry.
- Check for parting line seal.
- Check mould supports.
- Check parting line for contamination.
- Decrease size of vents if oversized, increase number of vents if necessary.
- Use a larger press if the machine is undersize.
- Reduce fill speed.

Fill to hold pressure sooner

PART STICKING IN CAVITY

Possible Solutions...

- Reduce mould temperature.
- Increase cooling time.
- Check for mechanical causes (limited draft, undercuts).

PART STICKING ON CORE

Possible Solutions...

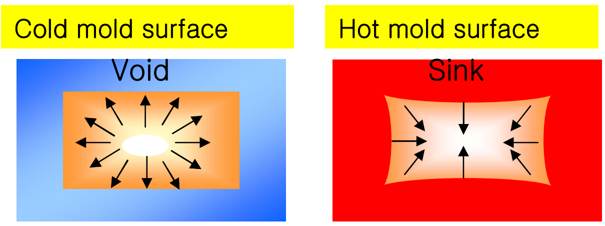
- Reduce injection pressure.
- Reduce mould core temperature.
- Decrease cooling time.
- Check for mechanical causes (limited draft, undercuts).

PARTS STICKING IN MOULD

Possible Solutions...

- Adjust mould temperature.
- Check ejector/puller design.
- Decrease injection rate / pressures.
- Decrease screw forward time.
- Increase draft angles on part.

SINK MARKS & VOIDS



Possible Solutions...

- Adjust feed to increase cushion.
- Decrease melt temperature.
- Adjust mould temperature.
- Check for a worn check valve assembly, barrel or screw.
- Decrease land length of gate and sprue.
- Decrease rib thickness to ~75% or less of adjoining wall.
- Differential mould temperatures; Lower mould surface temperature opposite rib side.
- Enlarge runner and gates to prevent freeze-off before part can be packed properly.
- Increase injection hold time, and /or pressure.
- Increase cooling time.
- Increase shot size.
- Reduce melt temperature.
- Provide additional mould vents.
- Relocate gates on or as near as possible to thick sections.
- Increase gate size.
- Increase pack pressure / time.
- Decrease the screw-back position if the cushion is too big.
- Eliminate heavy wall sections.
- Increase screw-back position if the cushion is too small.
- Increase injection pressure.
- Raise mould temperature.
- Reduce rib thickness.
- Relocate gate near heavy sections.

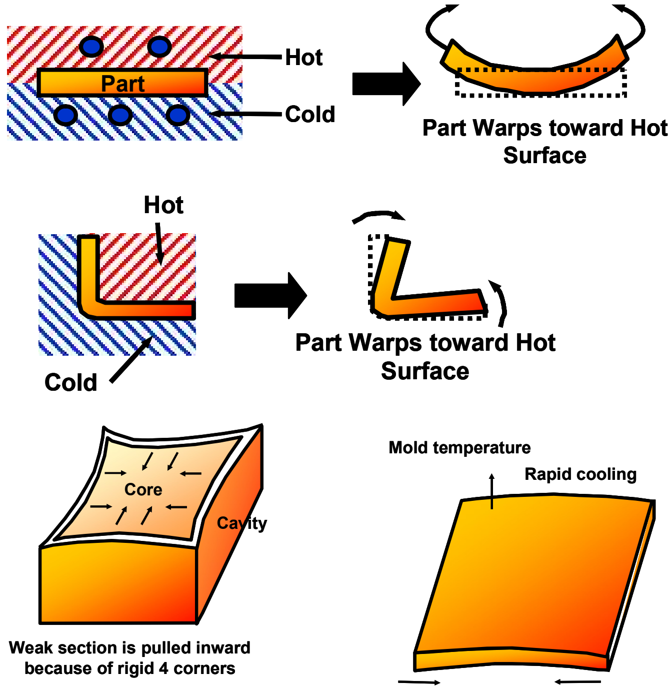
SPLAY MARKS/ SILVER STREAKING

Splay includes any defect that appears as smeared material on the part surface, usually oriented in the flow direction.

Possible Solutions...

- Be sure material is dried correctly.
- Adjust feed to increase cushion.
- Decrease melt temperature.
- Adjust mould temperature.
- Inspect check valve, barrel or screw for wear
- Decrease land length of gate and sprue.
- Decrease rib thickness to ~75% or less of adjoining wall.
- Reduce mould surface temperature opposite rib side.
- Enlarge runner and gates to prevent freeze-off before part can be packed properly.
- Increase injection hold time, and /or pressure.
- Increase injection pressure.
- Increase cooling time.
- Increase shot size.
- Reduce melt temperatures.
- Provide additional mould vents.
- Relocate gates on or as near as possible to thick sections.
- Increase gate size.
- Increase pack pressure / time.

WARPING



Possible Solutions...

- Adjust the gate dimension to improve flow of resin.
- Adjust the melt temperature to that recommended.
- Adjust the mould temperature.
- Adjust the injection speed and holding pressure.
- Be sure the gates are balanced on multiple gated parts.
- Check for parts hanging up in the tool on mould open and part ejection.
- Maintain uniform ejection.
- Correct mould temperature variations.
- Check mould temperature for hot spots.
- Equalise temperatures of both halves of the mould.
- Place parts on cooling jig during post moulding cooling.
- Increase packing time / cooling time.
- Review packing pressures.
- Reduce variation in wall thickness.
- Relocate gates to avoid thin sections, ribs or curved sections.

WELD LINES

When air gets trapped it often shows bubbles with foamy tails.

Possible Cause...

- Air entrapment (burning is the extreme).
- Inadequate venting.
- Fill pattern causing backfilling or flow front freezing and restarting.

Possible Solutions...

- Adjust gate, nozzle and sprue sizes.
- Change gate location to alter flow pattern or block one of the multiple gates.
- Decrease distance from gate to weld line by using multiple gates.
- Increase injection pressure and speed.
- Increase melt temperature and mould temperature.
- Increase the thickness of the weld line area or add a rib.
- Add or open vents in weld line area.
- Provide adequate venting in cavities and runner system.
- Vent cavity at point of weld/air entrapment

SKYGREEN® TECHNICAL SERVICE

Should you require any further technical information or assistance please do not hesitate to contact a member of the SKYGREEN® Technical Service Team – contact details below: