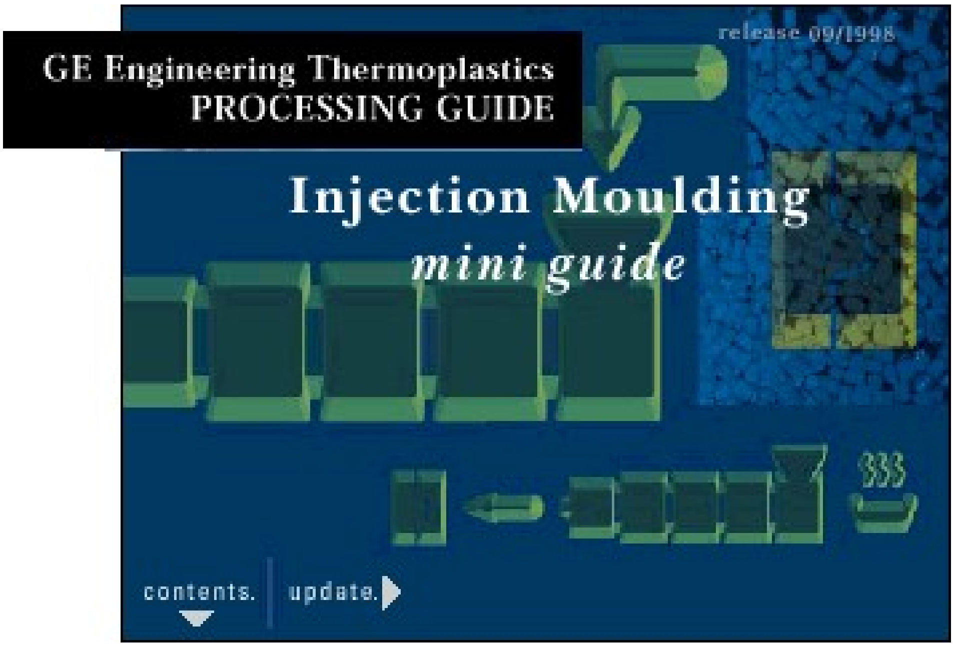
GE Plastics

Injection Moulding Mini Guide



This Injection Moulding mini guide is specially designed as a handy guide and quick reference for use on the shop floor. It includes details on pre-drying and temperature settings of machine, mould and material for all standard GE Plastics' resins.

Contents

Amorphous injection moulding resins

Cycolac®	ABS Resins
Cycoloy®	PC + ABS Thermoplastics Alloys
Lexan®	PC Polycarbonate Resins
Noryl®	PPE+PS Modified PPO® Resins
Noryl®Xtra	PPE+PS Modified PPO® Resins
Ultem®	PEI Polyetherimide Resins

Semi-crystalline injection moulding resins

Enduran®	PBT Thermoplastic Polyester Resins
Lomod®	Flexible Engineering Thermoplastic Resins
Noryl GTX®	PPE Modified PA Alloys
Supec®Xtra	PPS Polyphenylenesulphide Resins
Valox®	PBT+PET Thermoplastic Polyester Resins
Xenoy®	PC+PBT Thermoplastic Alloys

Other products

Azdel®- Azloy®-Azmet®	Technopolymer Structures
Remex®	Engineering Plastics
Kapronet	Purging Compound

Storage

GE Plastics' resins are supplied in the form of ready-to-process pellets in sealed bags or containers. No special storage conditions are necessary, however, all containers must be kept dry and away from sunlight since UV radiation could affect both packaging and contents. Special care must be taken to keep Lexan and Noryl GTX resins as dry as possible during storage.

When can resin pick up moisture?

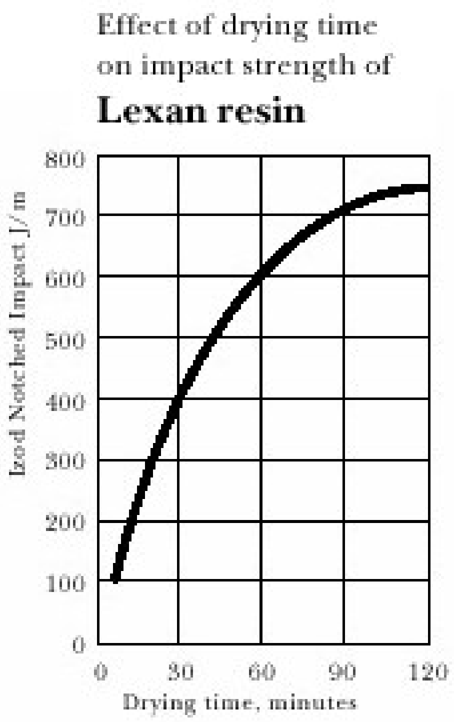
- during transport and storage
- during prolonged exposure to atmosphere, even after initial drying
- on its way to the machine hopper, when no hot (dry) air is used
- in case of high relative humidity

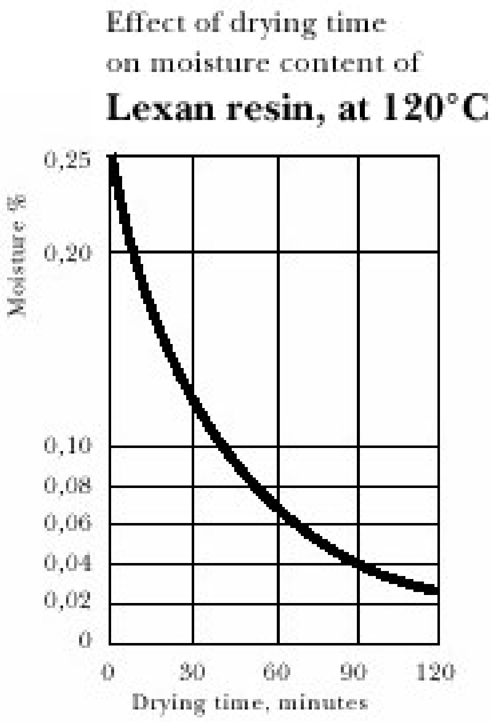
When can moisture be a problem?

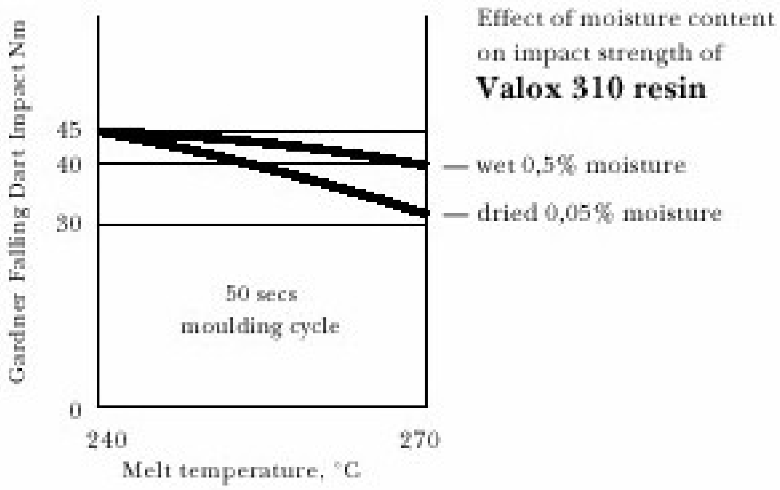
- pellets packed too deeply in the trays of drying ovens: lack of air circulation will prevent drying
- inefficient drying equipment
- machine hopper lid not sealed

Importance of pre-drying

Most thermoplastics absorb atmospheric moisture, which under normal processing conditions can cause degradation of the polymer. Specially Lexan, Ultem and Valox as hygroscopic resins demand proper pre-drying before moulding: these polymers do react with moisture at high (processing) temperatures. When this reaction occurs, polymer chains break, resulting in loss of properties. Excessive moisture can manifest itself in splay, silver streaking, blisters or degradation, which reduce the cosmetic and physical properties of moulded components. The removal of moisture is therefore essential to ensure optimal performance of the final part. Although Noryl resin has one of the lowest moisture absorption levels of engineering plastics, it is advised to dry before moulding - particularly where surface appearance is critical. For filled materials the drying time might be slightly longer; the allowed moisture level is the same. Due to a larger surface area of the pellets, recycled resins have a faster pick-up rate of moisture: drying time should in general be increased from 4 to 6 hours. It is recommended to dry regrind materials separately.







Pre-drying times

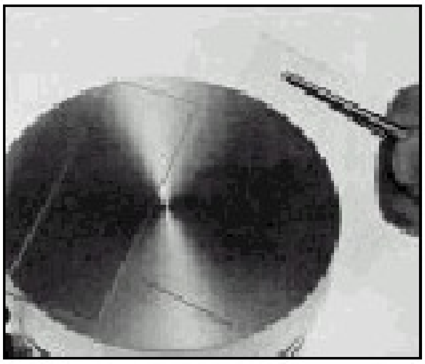
Although exceeding recommended drying times will not affect the properties of resins such as Lexan, Valox, Ultem and Xenoy, it is recommended not to dry for longer periods of time as indicated in this guide. With Noryl resin, exceeding drying times may result in surface defects due to oxidization. 90% of Noryl resin's moisture is removed anyway during the first hour of drying. Excessive drying time with Cycolac resin may give a slight surface oxidation of the pellets resulting in colour shift of light colours. This is caused by the fact that too long or too high air temperature may drive out required additives: resin will process poorly, parts get brittle, colours change. It is important that the typical drying times assume that the pellets are up to drying temperature 'before the clock starts'. Drying time taken to get the pellets up to drying temperature does not practically contribute to the effective overall drying.

For details per grade see datasheets.

TVI test for checking moisture content

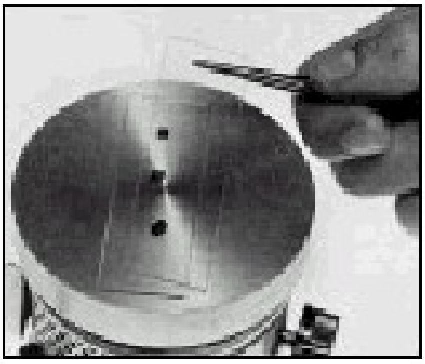
Measuring moisture content can be done by laboratory equipment such as the Carl Fischer titration method or others. Since these long, unwieldy lab techniques are impractical in most moulding shops, GE Plastics has developed a simple, fast and low-cost method to determine whether moisture-sensitive thermoplastic pellets are dry and ready for processing. In brief, this method means heating a few pellets to their melting point and observing whether bubbles are present (indicating moisture in the resin) or absent (indicating a dry material). Called TVI (Tomasetti's Volatile Indicator), the test requires little in the way of equipment and calls for just four simple steps. See pictures. The TVI test can be used for Lexan, Ultem, Noryl, Noryl GTX, Cycolac, Cycoloy, Valox and Xenoy resins, but not for glass reinforced grades. Not to be used for Supec and Lomod resins.

The necessary equipment is a hot plate capable of maintaining various temperatures from 200 to 350°C, two glass microscope slides, a straight edge or tongue depressor, a pair of tweezers and ideally a surface pyrometer to check the plate temperature - if no hot plate is available, a quick check can be done by using a heater band of the moulding machine. Moisture is denoted by bubbles in the flattened granules: the number and size increase with the amount of moisture present. A few small bubbles mean moisture contents of 0.02 to 0.03%, numerous bubbles indicate 0.05 to 0.1%, and many large bubbles moisture above 0.1%. To avoid misleading conclusions, it is advisable to always test at least four granules, as bubbles that appear in only some granules might be trapped air, rather than moisture.



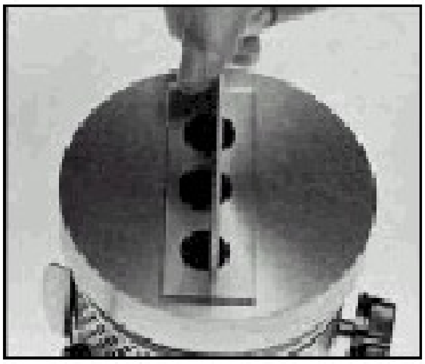
PICTURE 1

Heat two microscope glass slides on hot plate for about two minutes (be sure surface is clean)- Valox and Xenoy resins: 250°C, Noryl and Noryl GTX resins: 280°C, Lexan resin: 290°C, Ultem resin: 350°C, Cycolac resin: 250°C, Cycoloy resin: 275°C.



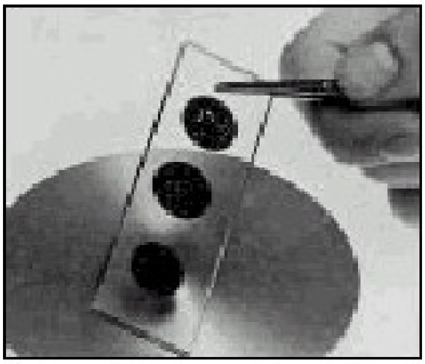
PICTURE 2

Use tweezers to place three or four pellets from the drier on one of the slides. Place the other slide immediately on top of the granules.



PICTURE 3

Press the two slides together with a straight edge until the granules are flattened out to about 12 mm diameter.



PICTURE 4

Remove the sandwiched slides and allow to cool. Amount and size of bubbles indicate percentage of moisture. If only one or two bubbles are present, they may be only trapped air and no moisture: to be sure, use at least three or four pellets.

Choosing a machine

Shot size %

· shot size%	- in general 30 to 90%, depending on resin type and in relation to residence time and melt temperature
· shot size less than 20 %	- can cause material degradation,part defects
· shot size > 90%	- no melt homogeneity, inconsistent part quality, packing problems

Residence time

· residence time	- always related to melt temperature
· residence time	- maximum 6 to 12 min. depending on resin type
· too long	- material degradation
· too short	- insufficient plastification and homogeneity of plasticized material
· colour pigments	- high residence times may change colours

Screw geometry and design

L/D ratio and compression ratio

· L/D 22:1 to 25:1	- for Valox, Xenoy resins
· L/D 18:1 to 24:1	- for all other resins
· CR 2:1 to 2.5:1	- for all resins
· CR's higher than 2.5:1	- not recommended

Non-return valves / back flow valves

· sliding ring valves	- for all resins
· ball check valves	- not recommended

Nozzles

· open nozzles	- for all resins
· reversed taper nozzles	- for Cycolac resin, not for Valox resin
· shut-off nozzles	- for Supec resin, not advised for other resins
· nozzle orifices	- as large as possible, 1 mm smaller than sprue top
· nozzle temperature	- heater band plus

	adequate control
· drooling	- check nozzle radius

Screw cushion

· screw cushions	- 3 to 10 mm of plasticizing stroke, dependent on screw diameter/material compression

Screw speeds

· r.p.m.	- always in relation to screw diameter
· circumferential speed	- most important
· circumferential speed	- unfilled and filled grades preferred 0.10 - 0.25 m/s.
· circumferential speed	- grades with special fillers preferred 0.10 - 0.20 m/s.
· r.p.m.	- adjust moulding cycle: screw rotation must fall in cooling time
· r.p.m.	- as low as practicable, but beware of insufficient plastification
· too high r.p.m.	- material degradationn
· too high r.p.m.	- breakage of glass fibres in glass filled products
· too high r.p.m.	- increased wear of screw and barrel
· too high r.p.m.	- inability to control melt temperature

Vented barrels

Do's

- do monitor carefully the temperature profile of the barrel
- do keep residence times below 8 min.
- do contact your local GE Plastics' representative for any advice on vented barrel processing

Don'ts

- don't process Ultem, Supec, Xenoy or Lomod resins using vented barrel equipment
- don't process FR grades using vented barrel equipment
- don't use vented barrel equipment for critical appearance applications
- don't change material frequently
- don't use filters in nozzles
- don't use – if possible – a vented barrel machine

Setting up production

Safety

General recommendations for safety in injection moulding are: appropriate protective clothing should always be worn when handling hot materials and during cleaning operations; continuous and direct removal of processing fumes and dust with a local exhaust system and a re-supply of fresh air will promote good workplace ventilation; avoid eye or skin contact with smoke, fumes, dust and vapours by installing an exhaust hood over the injection moulding machine nozzle. Similar ventilation precautions should be taken for all cleaning operations.

Purging change-overs

When changing materials during moulding, purging of the cylinder is required. Contamination with foreign or degraded resins can cause problems, such as delamination or black specks. Actual degradation of the to-be moulded material may result in parts which neither look nor perform satisfactorily.

Start-up from an empty and clean barrel is straightforward: barrel heaters are set according to the advised temperature profiles, and individual adjustments are made as production starts.

During breaks in production the following steps are advised:

1. close off hopper feed
2. empty plasticizing cylinder
3. move screw as far forward as possible
4. reduce barrel temperatures to recommended levels.

When starting up again, set the barrel heaters to normal processing temperatures, extrude until residual material is completely purged and begin moulding. The initial shots should be checked for contaminants in the moulded parts.

When a material change is necessary, or a production run ends, thorough purging of the cylinder is essential to remove all traces of previous polymers. Otherwise results may be unsatisfactory. Problems that may occur include material degradation, resulting in black specks, or, if materials are not compatible, delamination in the moulded parts. In some cases special procedures are necessary.

Purging can be done with nozzles either in place or removed, but removal allows cleaning and inspection for foreign matter. Be sure to avoid danger of splatter when purging. Add purging material with the screw rotating slowly. The purging material should be pressed out immediately and submerged in cold water before being disposed off. Purging should always be done with adequate ventilation.

Colours and FR

Extended interruption of production may result in melt dis-colouration: purging is necessary. Switching colours involves purging until the original colour is completely displaced. When using pigmented or flame-retardant materials, clean the screw and barrel mechanically. During reassembly of threaded components of the injection unit, care must be taken that no grease is in contact with the cylinder, screw surface or nozzle area.

Purging compounds

Cycolac resin	- for all resinsABS, PMMA, SAN
Cycoloy resin	- for all resinsABS, PMMA, PC, SAN
Lexan resin	- for all resinsPC, PP, PS
Lomod resin	- for all resinsPE, PP, PS
Noryl resin	- for all resinsPS, PMMA
Noryl GTX resin	- for all resinsPE, PP, PMMA

Noryl Xtra resin	- for all resinsNoryl, PS
Supec resin	- for all resinsHDPE, PP, GF PC
Ultem resin	- for all resinsGF PC, PC, PMMA, HDPE
Valox resin	- for all resinsPS, HDPE
Xenoy resin	- for all resinsPS, HDPE

Details per product

FR Cycolac resin

Production stagnation longer than 10 minutes or with normal shut down: purge with standard ABS. When changing to another ABS or another thermoplastic, all FR ABS must be removed before increasing heat settings. Since ABS is not particularly compatible with other polymers, standard ABS is the best purge material. When changing from a high viscosity thermoplastic to FR ABS, decrease temperatures and purge with FR ABS, then make temperature adjustment and continue moulding.

Cycoloy resin

Never switch off the heating when PC/ABS is in the barrel: reduce barrel temperatures to 160°C. Always purge FR Cycoloy.

Lexan resin

As with all PC containing thermoplastics resins,(Cycoloy PC/ABS resin, Xenoy PC/PBT resin), never switch off the heating when the resin is still in the barrel. Reduce to 170°C or clean screw with other material, otherwise black specks will occur or even the screw could be damaged; for transparent parts, it is strongly recommended to clean the screw thoroughly before starting moulding. Ideally a machine should only be used for moulding PC.

CHANGING FROM OTHER POLYMERS TO LEXAN RESIN

Thorough purging is especially essential before moulding Lexan resin in transparent, translucent or bright colours. Since the melt temperature required for Lexan resin is often higher than the degradation level of other thermoplastics, it is imperative that all traces of other polymers are purged out before starting moulding Lexan material. Extra care must be taken with POM. Its accelerated degradation at even the lowest melt temperatures, as advised for some Lexan grades, will produce unacceptable quantities of formaldehyde.

Contamination with nitrogen-containing polymers such as ABS and PA or FR polymers can cause chemical reactions that may severely degrade the polycarbonate, showing as dark spots in moulded parts. Should this occur after thorough purging of the machine, remove the screw and clean both cylinder and screw. A thorough inspection should be made of cylinder, screw and check rings for cracks and proper fit. Best purging material is polycarbonate or some proprietary compounds.

CHANGING FROM LEXAN RESIN TO OTHER POLYMERS

Where other materials are used after Lexan polymer, thorough purging to remove all traces of Lexan material must be done before the barrel has cooled. Purge with a PMMA and drop the cylinder temperatures if the resins to be moulded afterwards are POM, ABS or PA. Do NOT purge directly with PA or ABS after Lexan resin.

INTERRUPTION OF PRODUCTION

Keeping Lexan resin overnight or over a weekend in the cylinder is generally not recommended but can be done in the following way:

- reduce cylinder temperature to 170 to 180°C
- leave heaters on
- ensure that the temperature never drops below 160°C,otherwise Lexan resin will adhere to the cylinder walls and

may pull off metal particles and degraded resin as it cools and contracts. This contamination will show as black specks in the mouldings when production is restarted.

- during production delays, empty the screw to prevent overheating.

Lomod resin

Lomod resin requires no special purging procedures: normal purging with HDPE, LDPE, PP or PS will do. Because the processing temperatures of these materials are within the same range as used for Lomod resins, there is no need to reduce cylinder temperatures.

Noryl resin

PS and PMMA are effective purging materials for all Noryl resins. Purging should be done within the melt temperature range of the particular Noryl grade.

When moulding has to be stopped, the following steps are recommended:

- maintain cylinder temperature for interruptions of up to 15 minutes
- lower cylinder temperature by 40°C for periods from 15 minutes to 2 hours
- reduce further to 170°C for interruptions from 2 to 12 hours
- purge out barrel and shut off heat for periods longer than 12 hours

Noryl Xtra resin

Purge always with PS after process interruption.

Supec resin

HDPE, PP or glass-filled PC are suitable materials for purging the barrel after moulding Supec resin.

Ultem resin

CHANGING FROM OTHER POLYMERS TO ULTEM RESIN

Thorough purging is essential when changing to Ultem resins. Since the processing temperature of Ultem resins (350 to 410°C) is well above the degradation level of most other thermoplastics, it is essential to remove all traces of other polymers to avoid contamination resulting in black specks. Best purging material for Ultem resin is glass-filled Lexan regrind; drying is not recommended. Purge at the processing temperature of Ultem resin, continue to purge until the actual barrel temperature has reached 350°C, followed by the standard start-up procedure. Chemical purging compounds are not recommended. As with Lexan resin, take extra care with POM even at the lowest temperatures used for moulding Ultem resin. Its accelerated degradation produces unacceptable quantities of formaldehyde. FR polymers which contain nitrogen must be purged completely, otherwise contamination can cause chemical reactions that degrade Ultem resin.

CHANGING FROM ULTEM RESIN TO OTHER POLYMERS

Purging the cylinder after moulding of Ultem resin parts can be done by using Lexan polycarbonate. Put Lexan resin in the hopper when temperature settings are still at the high level as used for Ultem resin. The settings should be reduced to 300°C, however, the moment Lexan material is fed in. Purging can be completed by high viscosity PE or GE Plastics' Kapronet® purging compound. For stubborn cases purging can be done by using glass fibre-filled Lexan resin.

INTERRUPTION OF PRODUCTION

Short breaks: empty barrel and shut off hopper. Stopping production for more than 30 minutes: purge barrel empty and drop temperature settings down to 200°C. When stopping production for overnight, empty the barrel and drop temperature settings to around 200°C. If stopping for the weekend, switch off temperature settings after purging the barrel. When shutting down the machine, the hopper must be shut off at the throat and the machine should continue running until all residual resin is run out of the barrel. The screw must be left in its forward-most position with the barrel heaters set at 200°C for long periods of time. This reduces black speck contamination during start-up. If temperatures are dropped too low, Ultem resin will stick on metal parts and pull off particles, that result in black specks. Similar to Lexan resin.

Valox resin

When changing over to Valox resins from higher melt temperatures or heat sensitive materials, either PS or HDPE may be used as purging material. Due to the 'scrubbing' action of glass fibres, purging for glass-reinforced Valox resin may not be necessary.

Xenoy resin

Never switch off the heating when PC/PBT is in the barrel: reduce barrel temperatures to 160°C.

Preparation before setting

Preparation before setting

Part

- Was the part produced before? On what machine?
- Has the previous experience been recorded?
- How many mouldings are needed? By when?
- Any special measuring equipment needed?
- Weight and size of moulding: cylinder and clamping force?

Material

- Where is the material stored?
- Is the material pigmented? In what colour?
- Does the material contain recycle? And in what percentage?
- Does the material need pre-drying?
- Temperature and time of pre-drying?And with what equipment?

Mould

- Where is the mould stored? How can it be transported?
- What clamping equipment is needed?
- Does it fit in with the specified machine data, tiebars,core pulling devices, etc.?
- Are the mould and clamping devices ready to use?
- What temperature control is needed and what equipment does this require?

Machine

- When is the specified machine available?
- Is the machine in good working order, free from problems,lubricated, with clean mould platens etc.?
- What cylinder is needed? And what nozzle?
- Should the cylinder be purged?
- Has the additional equipment been planned: core pulling devices, air ejectors, robots etc.?
- Any trimming equipment, shrink fixtures needed?

Setting mould and machine

Setting mould and machine

Mounting the mould and setting the clamping unit

- set mould height on the machine with the clamping unit closed
- check clamping devices
- fix mould securely to the lifting tackle, ensuring that the two mould halves remain together, if necessary by bolting
- lock mould and thoroughly tighten up clamping bolts
- set clamping force at desired level
- set mould safety device: low pressure and high pressure
- set mould opening stroke, with damping.
- set ejectors
- set speed of the clamping unit
- set up the mould safety protection
- heat up mould to the required temperature

Setting the injection unit

- bring injection unit into rear position
- check nozzle radius and diameter
- check penetration depth of nozzle
- check the centring of the nozzle to the sprue bushing
- heat up cylinder
- set cylinder temperatures (not too high) and screw speed
- after having heated up the cylinder, set nozzle contact force
- make sure the mould is closed
- initially set material feed somewhat lower than the required shot weight
- set the position for change-over to hold-on pressure based on safety side
- plasticize stepwise under manual control, while observing how the material feeds: this should be consistent and not jerky movements
- after a short pause, purge out manually and check the melt temperature
- set injection speed and injection pressure to average values

Setting the machine controls

- set the operational selector switch to manual
- set injection time, after-pressure time and pause time initially somewhat longer than necessary, then optimize them after a few shots

Starting the injection process

When all settings have been checked, switch over from ‘manual’ to ‘semi-automatic’ mode. In general the settings must be optimized after several shots. Take following steps:

- increase or reduce metering
- optimize injection speed
- observe the material cushion in front of the screw
- check pressure build-up during injection
- check cooling time and if possible shorten it
- set screw r.p.m. to a low value: plastification time should be slightly shorter than cooling time
- check mould locking force
- harmonize machine movements to obtain a better balance of speeds, sequence and damping
- if visually acceptable mouldings are being produced,check quality
- where necessary, optimize and store the various functions
- record machine setting data
- ensure that mouldings are correctly taken out and transported away
- ensure quality control
- aim for a ‘fully-automatic’ machine cycle to ensure uniform moulding quality

Setting temperatures, speeds and pressures

Setting temperatures, speeds and pressures

Temperatures

Cylinder-, mould- and melt temperature settings as advised are guidelines, to be used for starting up the production. They are typical for most applications. However, where part geometry and mould design demand for it, higher melt temperatures can be used: the residence times at those temperatures should be kept as short as possible. Too high melt temperatures may result in colour changes when pigmented resins are used.

Injection speed

Injection speed is largely governed by the complexity of the part, the mould quality and the gating system. Slow injection speeds should be used during start-up.

However, fast injection speeds in general are desirable;they

- avoid premature freezing of the melt
- reduce visibility of weld lines
- give better welding of weld lines
- increase surface gloss
- keep melt temperature in the mould at a higher level
- improve transfer of pressure in the mould
- reduce overall orientation

Several effects can be created by changing the screw advance speed during injection.

A slower speed at the startof injection

- reduces mould deformation
- reduces tendency for jetting
- reduces matt spots in the gate area

A slower speed at the end of injection

- can reduce variation of properties of the moulding
- enables reduction of clamping force
- improves venting due to reduction of air compression

Profiled injection speed can control shear in some areas of the moulding, e.g. through grille areas etc.

Injection pressure

The injection pressure should be established to mould full parts consistently with a satisfactory finish - free of sink marks, weld lines or similar defects. The level of injection pressure, which should always be started at the minimum level, depends on many things:

- type and grade of material
- complexity of the part
- quality and polish of the mould
- type of gating
- material viscosity

Generally, the lowest pressures which still provide the desired properties, appearance and moulding cycle are preferred.

After pressure (hold pressure)

After pressure compensates for volume shrinkage of the melt during cooling in the mould. It should be maintained until the ‘gate is frozen’: the corresponding duration of after pressure should result in a constant part weight during moulding.

Insufficient holding pressure may lead to:

- sink marks
- voids
- variations in part dimensions
- increased shrinkage

Too high holding pressure may lead to:

- problems with ejection of the moulding
- stress in the gate area
- warpage

A decrease of the holding pressure at the end of the holding pressure time :

- reduces internal stresses
- reduces warpage because differences in shrinkage in the moulding far and close to the gate are reduced

Since after pressure is mould-related, depending on surface quality, part thickness, shape of gating, etc, it is difficult to recommend exact levels of after pressure.

Back pressure

Back Pressure has two functions:

1. it consolidates the melt by excluding air, thereby ensuring consistency of shot volume and removing splay caused by entrapped air.
2. it increases shear on the material: high shear may be necessary for masterbatching, dry colouring, mixing additives and sometimes to obtain a homogeneous melt with low-compression screws.

High back pressure is also used in structural foam moulding to prevent pre-blowing of the blowing agent. It can be used in a similar way for solid mouldings as a trick to reduce splay caused by (too high) moisture content.

The main function of back pressure is to improve mixing of the material: in most cases a low back pressure of 4 to 5 bar is sufficient to get a homogeneous melt without overheating the material. The maximum level of back pressure is related to screw diameter and screw r.p.m. Attention must be paid when moulding flame retardant, glass fibre-filled, pigmented resins and polymer blends. In general, levels of back pressure should not exceed 10 bar.

Mould temperature control

Mould temperature control

needs to

- heat up the mould to operating temperature - before production
- remove heat from mould cavity and control tool temperature- during production
- keep mould at uniform temperature - during production

results in

- optimization of cycle time
- insurance of high and uniform quality of mouldings

Mould temperature

- very important parameter
- controls cycle time
- controls product properties

surface appearance

dimensions

shrinkagee

warpage

crystallization rate

moulded-in stress levels

thermal behaviour of the moulded parts

- Amorphous materials

hot moulds

- improve properties
- no dramatic change of shrinkage values

cold moulds

- filling difficulties
- high injection pressures needed
- high melt temperature needed

- Semi-crystalline materials

hot moulds

- higher level of in-mould crystallization
- high shrinkage values

cold moulds

- crystallization after moulding
- danger of warpage during use of part

Water-based or oil-based units

- mould temperature below 100°C: water-based
- mould temperature above 100°C: oil-based or pressurized water-based
- heat transmission of oil is about half that of water: heat transmission surface area with oil is twice as big as for water
- pipes and couplings must take the heat requirements

Mould cooling system

- periodically clean entire temperature control circuit by flushing with a special solution: removes dirt and lime
- use insulating plates to insulate mould from machine
- reduces the time to heat the mould up to the required temperature
- extremely important when using hot-manifold and hot-runner moulds

Optimizing production

How to improve venting

Whenever a given mould creates problems with venting, such as burn marks or short shots, the first thing to do is to lower the injection speed. This does not increase venting, but it lowers the amount of air that has to leave the mould within a certain period of time. No burn marks on the part due to the diesel effect: as air cannot escape quickly enough, it will be compressed, its heat content will be concentrated in a small volume, resulting in a large temperature increase causing the incoming plastic to burn, (pressure x volume = constant). However, it could be necessary to increase the injection pressure in order to fill the part. If lowering the injection speed does not help, the venting should be increased. The amount of venting that is possible or required can be tested by taping very thin copper foil on the closing surfaces of the mould. Starting with 0.01 mm thick foil and gradually going thicker, it can be found by trial and error how much the venting can be increased before the part will show flash Attention when soft moulds are used: the copper foil may damage the mould surface.

How to improve mould release

Apart using well-known silicones-free release agents - not to be used when mouldings are painted or decorated afterwards - changing processing conditions can influence mould release. With beaker-shaped mouldings, it is important to keep the core lower in temperature than the cavity the part will shrink on the core and not stick in the cavity, where no ejector pins are available. This requires cooling the mould with separate fluid circuits. In general it is important to closely control the mould temperature. The temperature should not fluctuate too much, or increase in an uncontrolled way. A well-designed cooling layout, together with a correct way of connecting the various cooling channels - a ‘parallel’ layout rather than a ‘serial’ one - can help to keep the mould temperature as even as possible over the total mould surface. Overpacking - too high injection pressure, for too long - can create release problems: lowering injection pressure obviously is needed.

Too high mould temperature can lead to part warpage; too low mould temperature can result in insufficient flow, leading to overpacking. Increase of cooling time often has a positive effect. Although mould release is different for each GE Plastics’ resin, mostly high injection pressures and overfilling are the major causes of problems. A good way of checking is weight control of the parts during production.

How to influence mould shrinkage

Factors that influence shrinkage are cavity pressure and aftershrinkage. Cavity pressure is dependent upon mould temperature, melt temperature, injection speed, injection pressure, level of afterpressure and afterpressure time and, most importantly, the dimensions of gate- and runner systems (loss of pressure). Main parameters are injection pressure, afterpressure and afterpressure time. Parameters like back pressure, melt- and mould temperature have less influence on mould shrinkage. ftershrinkage depends on the temperature of the part when it leaves the mould. This temperature again At 23°C and 50% RH saturation can take months.

How to use hot runner tooling systems

Hot runners tend to make the whole process of moulding in general more susceptible to material degradation; not only due to longer residence times at higher temperatures, but also through the often not completely balanced flow/temperature control in a hot runner system. This may result in big differences in temperature in the various zones of hot runner blocks or nozzles.

Also very important is the lowering of the temperature of the hot runner when the moulding process is interrupted, (in case of problems for example), otherwise too much degradation of the material might occur. Purging the cylinder is rather easy, whilst purging the hot runner system is much more difficult. Sometimes however it may be necessary, otherwise the mould would be filled with degraded material. This might be difficult to remove from the mould surface and it could lead to staining. Also very important with hot runner systems is the option of an electrical circuit that allows gradual heating up. Eventual moisture present in the heater cartridges can evaporate: it avoids the chance of short circuiting. This is especially important in case a hot runner system has not been used for a long period of time. Also important is the possibility to heat the zones separately by using different control units. The power can be varied and thus uniform temperatures created.

How to save energy

Tips that not only save money, but also material:

- do not keep the machine nozzle always against the mould:after the gate is frozen, pull the cylinder back from the mould directly after plasticizing – in case of a shut-off nozzle
- use insulated plates between mould and machine
- aim for optimal control of the mould temperature; the allowed temperature difference between ‘temp-in’ and ‘temp-out’ of the coolant is a balance between quality of moulding (1 to 2°C) and economics (3 to 5°C)
- keep screw r.p.m. at a low level: the plastification time should be slightly shorter than the cooling time
- do not use a cylinder with a too high capacity for the actual shot weight

Quality control

Effective quality control enables simple yet accurate testing of materials and parts for behaviour and defects under processing conditions. Accepting the quality of the material, the main points are material degradation and unsatisfactory part performance.

Where materials have a narrow processing window, deviations in mechanical, thermal and physical properties can occur. The narrower this window, the harder it becomes to maintain part consistency, and thus rejection rates increase. Material that can be processed using a wide range of processing conditions provide an almost negligible rejection rate. This processing flexibility, however, makes the discovery of deviations from the optimum on moulded parts more difficult. Successful quality control is based on the comparison of production parts with samples of a good quality and known properties. Quality control systems based on statistical process models, such as SPC, are recommended.

For all parts, however, control methods should be related to the desired performance of the application, and, not least, the cost of it. The effects of costly engineering to attain certain unnecessarily high standards should always be borne in mind.

Visual control

This is the most important control method, as there is mostly a direct correlation between appearance and properties of the moulding. Extensive descriptions of deviations are discussed in ‘Part Defects and Corrective Actions’.

Mechanical control

Due to the varying shapes and sizes of moulded parts, mechanical testing can be difficult. It is therefore advisable to prescribe procedure, conditions and equipment carefully, and to experiment with testing. Mechanical testing demonstrates whether material quality and processing conditions affect the mechanical properties of the part, and can also be related to practical requirements. A common procedure is the falling dart test, which assesses the component’s ductility. The prime objective is to check for material degradation during processing. This affects impact properties, as do other deviations from optimal processing such as bubbles, sink marks, weak knitlines, etc.

Weight control

An economical, fast and easy method that can be carried out at the production location. Often preferred over dimensional methods, since weight variations will be more readily noticeable than those of dimensions. It can also assist in checking bubbles and voids, or other deviations from the filling rate of the cavity in the tool. Wide variations in weights of parts can indicate insufficient production and/or machine tolerances. Stabilizing part weight in general indicates stabilized processing conditions.

Dimensional stability

Factors such as orientation of polymer chains, internal stresses and filling rate of the cavity have an influence on performance characteristics of a part. Dimensional stability is thus dependent upon the control of all setting parameters on the moulding machine. Thermal stability tests can be carried out for shrinkage and/or deformation of parts following pre-conditioning at just above the resin's heat distortion temperature.

Complete mouldings should be used and a detailed study of each application is required to set correct parameters, using mouldings produced under optimal conditions.

Stress control

Due to tool or part design it is not always possible to produce really stress-free parts. For transparent materials such as Lexan resin it is often advisable to produce pre-production test mouldings: a light source with a linear polarized filter can then be used to detect internal stress concentrations. Stress levels can also be checked using stress corrosive fluids such as TnBP for Noryl resin and toluol N propanol for Lexan resin. (GE Plastics' representatives can supply more information.)

A simple but effective way to check stress in parts moulded out of Lexan resin is to observe them after 24 hours immersion in a 80°C solution of 100 g dishwashing powder with 10 g gloss agent in 1 liter water.

Viscosity control

Measuring viscosity of plastics materials can be used to check possible degradation of the resin resulting from injection moulding, UV ageing/weathering or heat ageing. Differences in flow 'before and after' indicate material degradation: any deviation from original granulate flow could be related to either 'abusive' moulding or non-controlled part ageing.

For Lexan PC and PC blends such as Cycoloy and Xenoy resins, the intrinsic viscosity or IV method is used. It demonstrates whether the material has been properly pre-dried and/or degraded during moulding. The percentage loss in IV of the material due to moulding of the part is an indication of material degradation. Information on exact figures can be obtained via GE Plastics' representatives. For all materials the MVR (melt volume rate) or MFR (melt flow rate) can be used - same test, just a different way of measuring the result: MVR in cm 3 /10 min., MFR in g/10 min. The delta flow data of MVR and MFR show the change of the material during and after processing. During processing, phenomena such as thermal degradation, crosslinking and other desired or undesired processes can occur. These are very dependent upon moulding conditions, part design and tool design.

Other methods

Very reliable methods of quality control are tests according to government and agency standards such as VDE, KEMA, SEMKO, NEMKO, etc. For these standards, test methods for both application and material are provided. This avoids possible confusion between moulder and end-user. More and more computer-based control systems are used nowadays. Many new software and hardware tools enhances the quality of moulds, machines and mouldings produced with them. They give possibilities to optimize, visualize and monitor basic settings of the injection moulding machine. Using pressure sensors in moulds, filling phases can be analyzed to get information about orientation, appearance and crystallinity of moulded parts. Deviations of several shots following each other can be calculated statistically and monitored afterwards. Differences in quality can be signalled immediately.

Fault diagnosis

If a part has been produced satisfactorily and it goes wrong,something has changed. The principle is not to change conditions immediately. The causeshould beidentifiedandrectified.

Never adjust one condition to compensate for a failure or change in another condition. For example, if melt temperature has increased,don'treduce speeds or pressures or mould temperature to compensate.Identify problemthat is to say reduce melt temperature to what it was before, by checking and replacing thermocouples, etc.

Ask questions

- what haschanged
- what is fault?
- when did it start?
- how often does it occur?
- where is fault?
- is the fault randomly situated or always in same place?
- etc.

Identify causes of defects

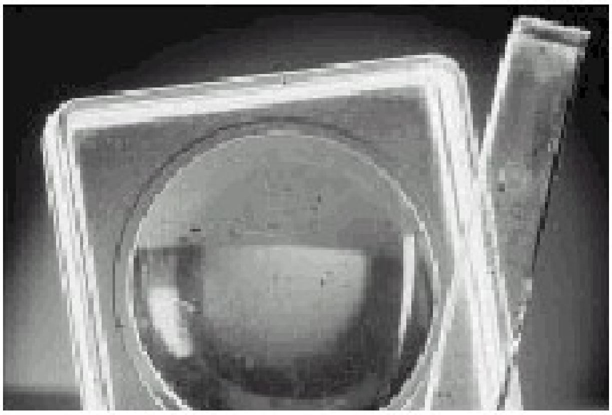
- test, observe, conclude and study ‘history of faults’
- injection speed: test
- screw speed: test
- back pressure: test
- melt temperature: test
- etc.
- machine: check
- heating of cylinder
- mould temperature control
- material cushion

- locking of mould
- etc.

Fault descriptions, causes and actions

Black specks

Dark spots due to thermal damage



DESCRIPTION

Small black areas (spots) inside the material,mostly present in transparent resins.

CAUSES

machine

- down time too long
- barrel switched off over a long period of time
- poor purging of barrel
- dirty plasticizing unit
- inadequate nozzle

mould

- dead edges in gate/runner system

material

- granule impurities
- degradation by other resins
- pick-up of degraded material from cylinder wall during cooling

ACTIONS — IN ORDER AS SHOWN
1. purge with an appropriate material

in general: Kapronet

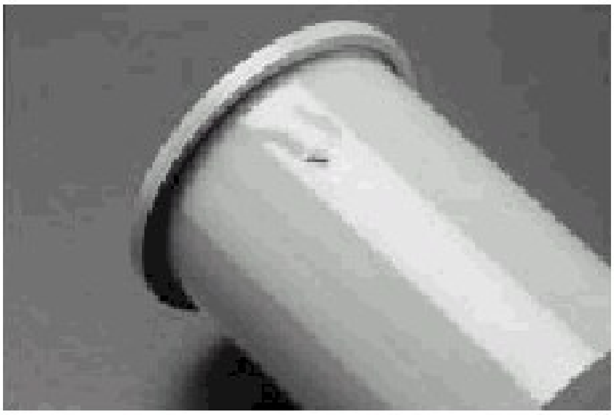
for Lexan PC: ground acrylic or regrind Lexan resin

for Ultem resin: regrind Ultem resin or glass-filled Lexan

resin – don’t drop temp. settings while purging;
2. check for impurities: use uncontaminated material,do proper housekeeping
3. check and adjust melt temperature
4. check for dead edges: nozzle, back flow valve, gates/runners
5. check for screw wear

Blisters, bubbles

Blisters can cause bulges on the surface



DESCRIPTION

Small air-or gas-filled hollows in the moulding,cooling voids.

CAUSES

machine

- injection pressure too low
- inadequate functioning of back flow valve
- suck-back too long
- plasticizing too fast
- air trap in the hopper feed
- improper feed

mould

- volatiles and trapped gas
- mould temperature too low
- thin to thick transition

material

- melt overheating

ACTIONS — IN ORDER AS SHOWN
1. control holding / injection pressure
2. increase back pressure
3. increase mould temperature
4. check back flow valve
5. allow for adequate venting
6. enlarge gate
7. shorten land length

Blush / flow marks

Dull spot (corona) near the sprue

DESCRIPTION

Blush and flow marks are the result of variations in material temperature, caused by the temperature gradient between machine nozzle and mould sprue bushing. A halo around the direct sprue is the result of cold material in the nozzle tip section. Eliminating the temperature gradient will minimize the blush and halo effects.

CAUSES

machine

- injection speed too slow or too fast
- injection pressure too low
- hold pressure too long

mould

- inadequate mould cooling
- mould too hot around gate
- mould too cold
- gate too small
- wrong gate location
- gate land length too long
- wrong hot runner system

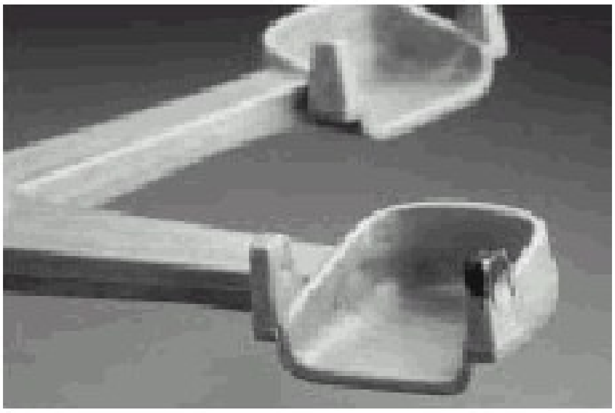
material

- melt temperature too low

ACTIONS — IN ORDER AS SHOWN

1. adapt injection speed
2. add a large cold slug area
3. add cold wells at the end of runner systems
4. control nozzle heat better: if necessary add beryllium copper tip (not recommended for FR resins)
5. shorten or eliminate standard sprue bushing:use a hot sprue bushing
6. clean flow must exist from the cylinder, adaptor, nozzle and tip: avoid and eliminate any dead pockets or sections

Burn marks / diesel effect



Diesel effect (burns) due to entrapped air at the end of the flow path

DESCRIPTION

Burn marks are (often) brown streaks. They are usually caused by overheating the material due to entrapped air (diesel effect): this causes the darkening in colour.

CAUSES

machine

- problems with back flow valve
- injection speed too fast
- back pressure too high

mould

- inadequate venting : entrapped air
- frictional burning
- check sprue diameter

material

- melt too hot or too cold:may create shear

ACTIONS — IN ORDER AS SHOWN

1. check venting channels for dirt
2. decrease injection speed
3. decrease injection pressure
4. use programmed injection
5. check for heater malfunctioning
6. reduce screw r.p.m.
7. decrease nozzle temperature
8. reduce melt temperature
9. improve mould cavity venting
   - add vents to ejector pins
   - add vents to parting line of part
10. enlarge gate to reduce frictional burning
11. alter position and/or increase gate size

Delamination

Cosmetics article with flaked off 'skin'

DESCRIPTION

Separation of layers in the moulded part that can be peeled off: flaking of surface layers. It results from insufficient layer bonding due to inhomogeneities and high shear stresses.

CAUSES

machine

- injection speed too high

mould

- mould too cold
- sharp corners in gate area
- shear heat caused at sharp corners

material

- melt too hot
- poorly melted
- incompatible colour dyes
- cross contamination with other polymers
- too much use of recycled material

ACTIONS — IN ORDER AS SHOWN

1. increase melt temperature
2. increase mould temperature
3. decrease injection speed
4. eliminate contamination
5. check percentage regrind
6. dry material
7. purge equipment
8. change material
9. radius all sharp corners at gate

Dimensions of part

DESCRIPTION

Excessive shrinkage means part dimensions differing from expected ones; amorphous resins behave differently to semi-crystalline materials.

CAUSES

machine

- injection pressure too low
- injection hold pressure time too short
- overall cycle too short
- back flow valve cracked
- excessive cylinder clearance
- heater bands burned out

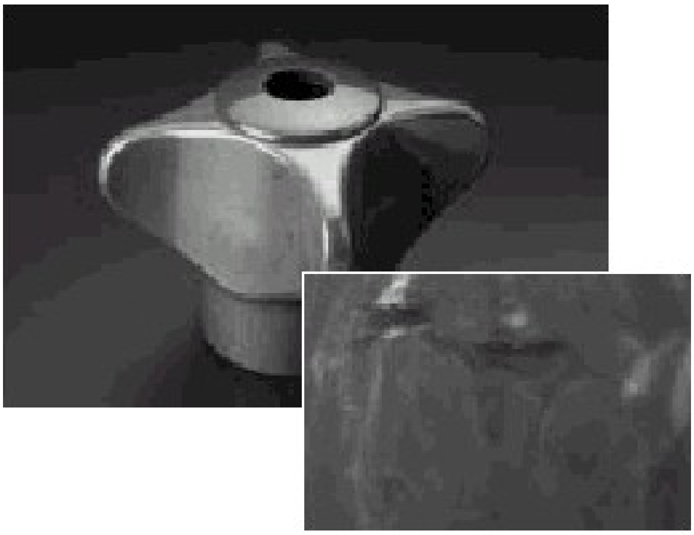
mould

- too hot
- gates too small (related to inadequate pressures)
- gating in the wrong place
- mould too small

ACTIONS — IN ORDER AS SHOWN
1. increase injection pressure
2. increase cooling time
3. increase mould temperature
4. maintain uniform cycle time operation
5. check machine for erratic operation
6. check the percentage regrind to virgin material
7. increase gate size
8. reduce gate land length
9. relocate gate if glass-filled compounds
10. balance runner and/or gate system
11. reduce number of cavities to balanced system

Discolouration

Colour streaks due to poor blending in the plasticizing unit



DESCRIPTION

The appearance of areas in the moulding with a deviating colour.

CAUSES

machine

- contamination

mould

- check sprue diameter
- pin-point too small
- poor venting

material

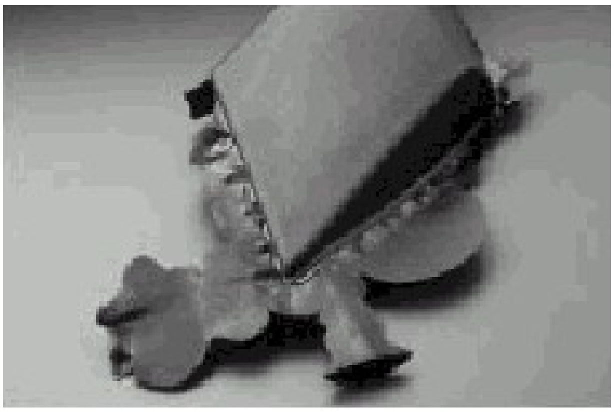
- melt too hot or too low:may create shear
- long residence time
- instability of polymer/pigments

ACTIONS — IN ORDER AS SHOWN

1. purge heating cylinder
2. lower material temperature by
   - reducing cylinder temperature
   - decreasing screw speed
   - reducing back pressure
3. lower nozzle temperature
4. check residence time
5. check machinery purging
6. shorten overall cycle
7. check hopper and feed zone for contaminantst
8. check for proper cooling of ram and feed zone
9. provide additional vents in mould
10. move mould to smaller shot size press to reduce residence time

Flash

Large area overspraying



DESCRIPTION

A film of material attached to the moulding at the mould parting line.

CAUSES

machine

- clamping pressure too low
- injection pressure too high
- injection speed too fast

mould

- inadequate mould supports
- clamping force too low
- damaged mould surface: parting line

- excessive projected area

material

- melt too hot
- viscosity too low

ACTIONS — IN ORDER AS SHOWN

1. reduce injection speed
2. reduce injection pressure and/or booster time
3. increase clamping force
4. check mould for proper mould support and/or parallelism
5. reduce melt temperature
6. reduce mould temperature
7. check excessive vent depths
8. change to higher clamping machine

Jetting

Jetting starting at the gate, spreading over the entire moulded part

DESCRIPTION

A turbulent flow in the resin melt: the melt strand enters the cavity in an uncontrolled way. Due to cooling down, the strand is not fused homogeneously with the melt that follows. It shows as a serpentine line on the part surface. Too little restriction when filling the cavity: material is injected in empty space (wrong gate angle).

CAUSES

machine

- injection speed too fast

mould

- mould too cold
- gates too small
- gate land length wrong
- wrong gate location

material

- melt too cold

ACTIONS — IN ORDER AS SHOWN

1. decrease injection speed
2. check nozzle heating
3. increase mould temperature
4. increase melt temperature

5. increase gate size
6. avoid gating at thick section
7. modify gate location or angle: directly into wall or pin
8. use tab gate or submarine plus pin

Pitting

DESCRIPTION

Pitting is the presence of unmelted particles due to difficult dispersion of additives, wrong mixing or cross-linking during processing.

CAUSES

machine

- wrong or worn out screw, giving hang-ups
- melt temperature too low
- injection pressure too low

mould

- shear in gating
- sharp corners

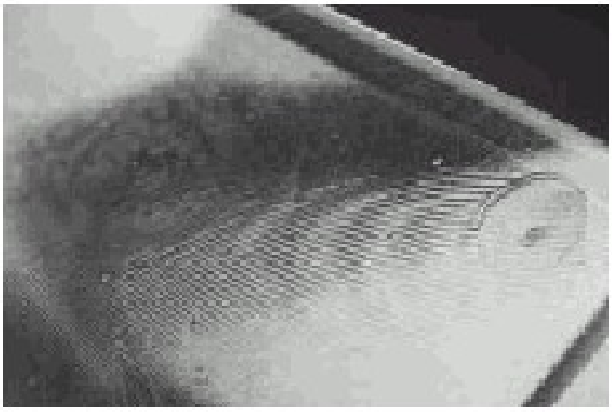
material

- inhomogeneous material
- contamination

ACTIONS — IN ORDER AS SHOWN
1. get shear down
2. lower back pressure
3. decrease injection speed
4. change temperature profile
5. check regrind percentage
6. check shot size vs part
7. check hot-runner: torpedoes

Record grooves

Concentric record grooves



DESCRIPTION

Resembles the grooves of gramophone records. At slow speed - as material nears cold tool - it loses its flow (below HDT) before actual contact. The melt that follows flows over cooled melt, to repeat the cycle.

CAUSES

machine

- injection speed too low
- injection pressure too low

mould

- mould too cold
- different texture of mould halves: polished vs coarse grain

material

- melt too cold

ACTIONS — IN ORDER AS SHOWN
1. increase injection speed
2. increase injection pressure
3. increase melt temperature
4. increase mould temperature
5. check mould textures

Release problems

Sticking in cavity

DESCRIPTION

At end of cycle, the moulding does not release from the mould but sticks in the cavity (female mould side).

CAUSES

machine

- injection pressure too high
- injection speed too high
- holding time too long
- too much material feed

mould

- cavity too hot – release is better from hot mould (20°C below HDT)
- mould too cold
- poor mould finish

material

- melt too hot

ACTIONS — IN ORDER AS SHOWN
1. check cycle time: cooling
2. decrease injection pressure
3. decrease injection hold time
4. decrease injection speed
5. decrease booster time
6. reduce and adjust feed for constant cushion
7. check for poor mould finish or corrosion on mould surface
8. increase mould opening time
9. decrease material temperature by lowering cylinder

10. lower mould temperature
11. adjust the cavity temperature to a 20°C differential between mould halves
12. check mould for undercuts and/or insufficient draft and taper
13. use proper mould release

Sticking on core

DESCRIPTION

At end of cycle, the moulding does not release from the mould but sticks on the core (male mould side).

CAUSES

machine

- injection pressure too high

mould

- core too hot
- core bending
- creation of vacuum — especially on thin-walled parts

ACTIONS — IN ORDER AS SHOWN

1. check cycle time: cooling
2. decrease injection pressure
3. decrease injection hold time
4. decrease booster time
5. adjust feed for constant cushion
6. decrease mould-closed time
7. increase core temperature
8. decrease cylinder and nozzle temperature
9. check mould for undercuts and/or insufficient draft
10. check mould for bending: rule of thumb is 1:5 for core diameter to core length

Sticking of sprue

DESCRIPTION

At end of cycle, the sprue does not release from the mould but sticks in the sprue bushing.

CAUSES

machine

- injection pressure too high
- too much material feed
- nozzle frozen
- nozzle diameter too large for sprue bush
- inadequate draft angle
- drool from nozzle

mould

- too hot
- ineffective sprue pullers

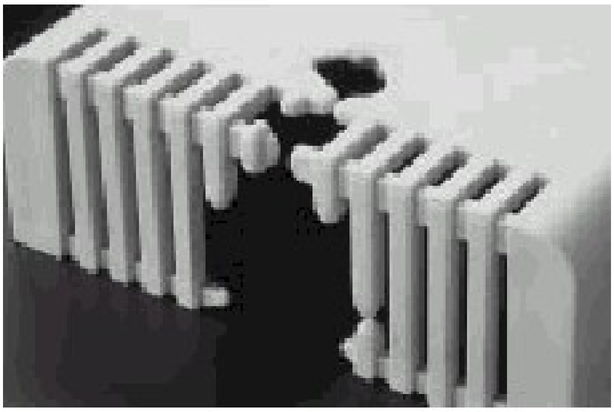
ACTIONS — IN ORDER AS SHOWN

1. decrease injection pressure
2. decrease injection hold time
3. decrease booster time
4. increase die-closed time
5. decrease mould temperature at sprue bushing
6. leave nozzle against mould: no pull back
7. raise nozzle temperature
8. check incorrect seat between nozzle and sprue: sizes and alignment of holes in nozzle and sprue bushing
9. sprue bushing hole must be larger: reduce nozzle diameter for sprue bushing being used
10. check polishing of sprue
11. check proper design of sprue puller pin
12. check cone of sprue: usually 1:20, for difficult jobs 1:15
13. provide more effective sprue puller:

- increase sprue puller by increasing taper of sprue puller
- polish worn or rough sprue bushing

Short shots

Solidified flow front on a glass fibre reinforced housing



DESCRIPTION

Resulting from incomplete filling of the mould: parts of the moulding are not formed.

CAUSES

machine

- improper feed
- injection pressure too low
- injection speed too low
- injection time too short
- faulty back flow valve ring

mould

- poor mould venting
- mould too cold

material

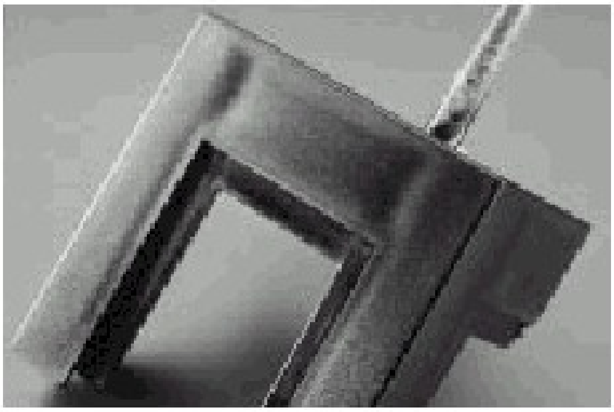
- melt temperature too low
- viscosity too high

ACTIONS — IN ORDER AS SHOWN
1. increase dosage
2. increase injection pressure

3. increase booster time forward
4. increase material temperature by increasing cylinder temperatures
5. increase mould temperature, if glass-filled
6. check material flow length vs wall section thickness
7. increase nozzle diameter
8. check restrictions of nozzle, runners and actual gating
9. increase gate size of sprue and runner system

Sink marks

Sink marks due to wall thickness variations



DESCRIPTION

Visible defects resulting from insufficient cooling before removal from the mould. A heavy rib intersecting a thin wall may show up sink marks: these are very difficult to eliminate by varying processing conditions. Too high holding pressure - useless when gating is too small - creates very high stresses in gate areas.

CAUSES

machine

- injection pressure too low
- injection pressure time too short
- short of shot capacity

mould

- mould temperature too high: too high shrinkage
- gate too small: leads to early cooling/freezing at the gate, holding pressure cannot help anymore to compensate for the shrinkage
- land length too long
- wrong dimensions rib vs wall

material

- melt too hot

ACTIONS — IN ORDER AS SHOWN
1. increase injection speed to maximum range
2. sometimes lower injection speed: crystalline materials
3. increase injection hold time
4. increase injection pressure
5. reduce melt temperature
6. reduce mould temperature
7. check for hot spots: separate water channels in cooling system / add heat pipes such as thermal pins or beryllium copper slugs for spot cooling
8. enlarge and/or add vents to mould parting line
9. increase size of sprue and/or runner

10. increase gate size and reduce gate land length
11. relocate gate next to heavy or thicker areas
12. core out heavy wall sections where possible
13. incorporate textured surfaces

Splay / streaks

Splay

DESCRIPTION

Splay marks, silver streaks, splash marks are the result of
(a) moisture on the pellets which should be removed under recommended drying times and temperatures
(b) products of degradation due to overheating
(c) residual non-aqueous volatiles in material
(a) and (c) will produce fine lines emanating from the gate all over the part whereas (b) will show up as coarse lines, lumps in sections of the moulded parts.

CAUSES

machine

- degraded material

hot spot in cylinder

material hang-up area at nozzle tips or adaptors

- injection pressure too low or too high
- injection speed too low or too high
- back pressure too low

mould

- frictional burning at gate, in machine nozzle or hot runner
- trapped volatiles

material

- melt too hot
- contamination in resin
- excessive moisture
- Noryl resin: degradation of material due to too long pre-drying at high temperatures

ACTIONS — IN ORDER AS SHOWN
1. check pre-drying: dry material before use
2. check moisture content after pre-drying
3. check effectiveness of drying equipment: temperature and time
4. lower nozzle temperature
5. lower material temperature by:

- lowering cylinder temperature
- decreasing screw speed
- lowering back pressure

6. decrease injection speed
7. raise mould temperature
8. shorten or eliminate screw decompression
9. shorten overall cycle
10. increase back pressure; in case of drooling,reduce back pressure
11. check for drooling

12. check for contamination(e.g. water or oil leaking into mould cavity)
13. barrel purging (hang-ups)
14. allow for adequate venting
15. open gates
16. move mould to smaller shot-size press

Gate splay

DESCRIPTION

Gate splay is the appearance of dull spots around the gate, resulting from temperature differences in the material, (too high shear forces tearing the surface layer). Often moisture streaking resulting from improper pre-drying.

CAUSES

machine

- injection too fast

mould

- mould too cold
- gate too small

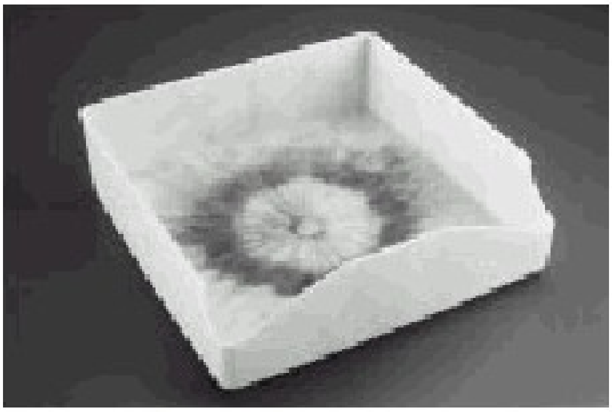
material

- melt too cold

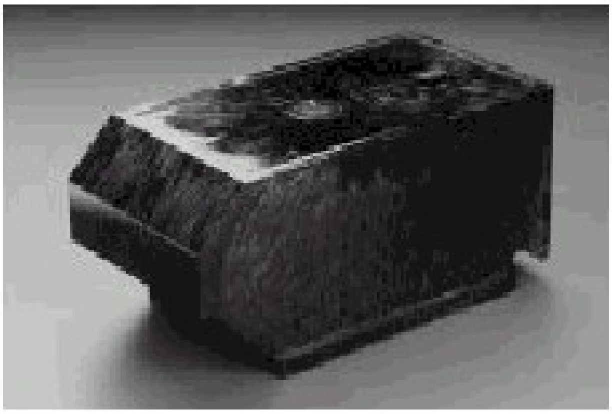
ACTIONS — IN ORDER AS SHOWN

1. decrease injection speed
2. increase mould temperature
3. increase melt temperature
4. increase gate size
5. change gate location

Streaking

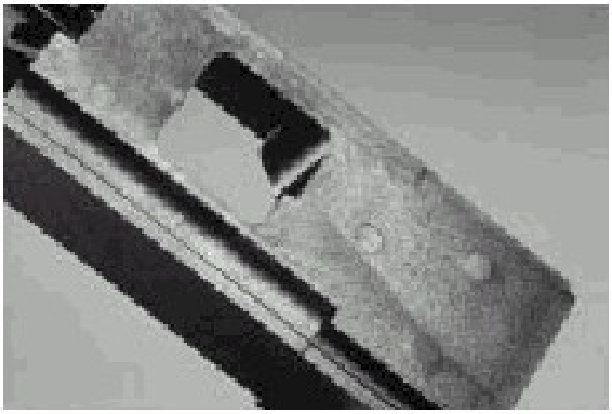


Burnt streaks due to excessive residence time in the plasticizing cylinder



Streaks due to excessive moisture content of the granules

Air streak (near the sprue) due to sucked in air during decompression



Glass fibre streaks (clearly visible weld line)

DESCRIPTION

The appearance of large, dull and lamellar areas on the surface of a moulding.

CAUSES

machine

- damaged back flow valve ring

mould

- areas of hang-up
- hot spots

material

- contamination caused by stock or machine

if steady pattern: machine

if erratic pattern: material pigmentation /

instability of material

ACTIONS — IN ORDER AS SHOWN
1. check for material contamination by other resins
2. check barrel purging
3. check for cracked or worn back flow valve ring
4. check for worn feed screw
5. check for excessive clearance on screw/barrel dimensions
6. check for overheated cylinder heater bands
7. check for overheated nozzle heater bands

Stringing

DESCRIPTION

Stringing is the appearance of a thin plastic string coming from the sprue.

CAUSES

machine

- back pressure too high
- nozzle temperature too high

mould

- wrong sprue

material

- insufficient melt strength

ACTIONS — IN ORDER AS SHOWN
1. use suck back: only for crystalline materials
2. lower back pressure
3. lower or increase nozzle temperature
4. use different temperature profile
5. do not use sprue breaks

Voids

DESCRIPTION

Vacuole hollows (‘empty bubbles’) in the moulding,due to thermal shrinkage that draws material away from the fluid core of a part.

CAUSES

machine

- injection pressure too low
- injection pressure time too short
- injection speed too high
- back pressure too low

mould

- mould temperature too low
- incorrect material flow
- part wall too thick

material

- melt too hot
- wrong material grade: some Lexan resin types more sensitive

ACTIONS — IN ORDER AS SHOWN
1. decrease injection speed to medium range
2. increase holding time
3. reduce melt temperature
4. increase mould temperature
5. check gate size: too small results in freezing at gate with voids and sinks in other areas of the part
6. increase gate size and reduce gate land length
7. increase nozzle size and/or runner system
8. redesign part to obtain equal wall sections

Warpage, part distortion

DESCRIPTION

A dimensional deformation of the moulding resulting from frozen-in stress, or because the part was taken too hot from the mould. Basically it is due to pressure differences between areas.

CAUSES

part

- wrong part design
- part too heavy

machine

- insufficient cooling time
- too high injection pressure

mould

- wrong gate location:different shrinkage in different flow directions
- too big undercuts
- inadequate ejector pins
- cavity too hot

material

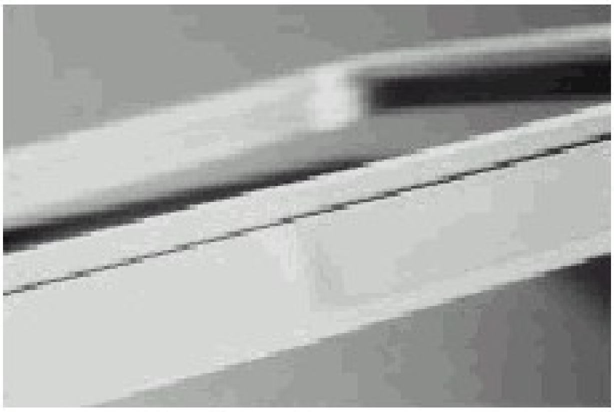
- orientation of fillers
- wrong material choice

ACTIONS — IN ORDER AS SHOWN

1. equalize temperature of both mould halves
2. observe mould for uniform part ejection
3. check handling of parts after ejection from mould
4. check part weight: take care with Valox resin
5. increase injection hold time
6. increase cooling time
7. increase or reduce injection pressure
8. increase mould close time
9. increase or reduce mould temperature
10. set differential mould temperatures to counteract warpage due to part geometry
11. use shrink fixtures and jigs for uniform cooling of the part
12. check gate locations and total number of gates to reduce orientation
13. additional gates may be required to overcome overpacking or underpacking on large parts
14. increase gate dimensions
15. change gate location if glass-filled, notice fibre orientation
16. redesign part to equalize wall variation in moulded part – heavy and thin walls in same part create differential shrinkage stresses

Weld lines / knit lines

Colour differences at a weld line



DESCRIPTION

These lines occur where two plastics flow fronts in the mould meet. The streams of plastic should be hot enough to fuse adequately. Weld lines are not just surface marks, but can be points of weakness: notches, stress raisers.

CAUSES

part

- wrong part design

machine

- injection speed too slow
- injection pressure too low
- injection time forward too short

mould

- mould too cold
- insufficient venting
- inaccurate functioning of back flow valve
- distance from gate excessive

material

- melt too cold

ACTIONS — IN ORDER AS SHOWN
1. increase injection pressure
2. increase injection hold time
3. increase injection speed
4. raise melt temperature by increasing cylinder temperatures
5. raise mould temperature
6. check for proper venting of the part
7. vent the cavity in the weld area
8. provide an overflow well next to the weld area
9. change gate location to alter flow pattern
10. increase gate and/or main runner system
11. reduce gate land length
12. spot heat particular area with thermal pins or cartridge heaters
13. use textured surfaces

Reusing Materials

Thermoplastics resins in general can be recycled into similar applications: this means that a company can grind, clean and eventually upgrade the material. GE Plastics' product range of selected post-consumer recycle materials is called Remex Engineering Plastics. This activity of reusing materials is called'recycling'.

During production, such as with injection moulding, sprues, faulty mouldings, short shots, and so on, can be reground directly in the production facility and reused. This can be done by using reground resin on its own, or by mixing regrind with virgin material. This activity of reusing materials is called 'regrinding'To avoid misunderstanding, only'regrinding'will be discussed here.

Sprues, runners and faulty mouldings produced from GE Plastics' resins can all be reground. Care should be taken to ensure that reground material is not degraded and is clean and free from impurities. Parts that show traces of over heating or burning, and also parts with humidity related defects such as splash, should never be reground. Obviously reground materials should be of the same composition and not be mixed with other plastic grades.

If there is the slightest doubt whether degradation or contamination has occurred, the material should not be reused. It is also very important that reground materials are properly pre-dried, as with virgin materials.

Although many resins show minimal reduction in properties after regrinding, special attention is required to ensure that reground materials in principle are not used for impact critical applications. Care must also be taken when reusing flame retardant or heavily pigmented materials.

In general, it is very difficult to quantify what mixing percentages of reground: virgin material are possible. These differ from case to case, being related to customer-defined part quality requirements and applicable standards and regulations.

The following factors should be borne in mind when using reground materials:

- moulding conditions of first moulding process
- impact behaviour of final parts
- flame retardancy, UL requirements
- UV resistance
- colour stability
- chemical resistance
- physical properties
- preparation of material prior to use

- disposal of dirty parts
- proper pre-drying
- moulding conditions of second moulding process

GE Plastics understands that the plastics processing community wants to take advantage of this important feature of thermoplastics materials. Therefore, while the company can only be held liable for the quality of delivered virgin material, it is committed to supplying the information which will allow processors to achieve the levels of quality and performance expected by customers. GE Plastics recommends the use of a specific methodology wherever plastics are to be reground for reuse.

The recommended methodology to determine whether, and to what extent, reground materials can be used for the production of industrial products is as follows:

- define the critical characteristics and the acceptable level of performance in agreement with the final end-user
- conduct trials with different mixing levels of regrind,for example in steps of 10, 20, 30%, etc.
- measure the characteristics of the parts produced with these levels
- compare the obtained values with the acceptable level of performance and select accordingly
- agree with the end-user the maximum acceptable proportion of regrind