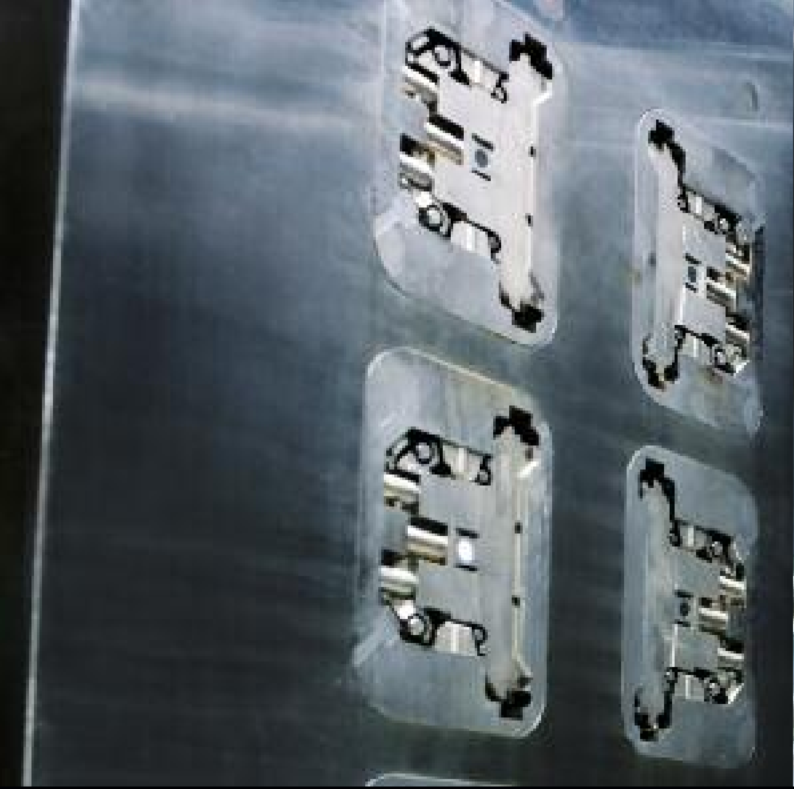


PROCESSING DATA FOR THE INJECTION MOLDER

Order No. LXS-SCP-006EN, Edition: 2009-01

LANXESS DEUTSCHLAND GMBH
BUSINESS UNIT SEMI-CRYSTALLINE PRODUCTS
D-51369 LEVERKUSEN | WWW.LANXESS.COM
WWW.DURETHAN.COM | WWW.POCAN.COM

LANXESS
Energizing Chemistry

LANXESS
Energizing Chemistry

CONTENTS PAGE

1. PRODUCT OVERVIEW

The purpose of this brochure is to give the injection molder a quick rundown on the processing of the engineering plastics produced by LANXESS:

Durethan®	**PA 6, PA 66, PA 6I, Co-PA**
Pocan®	**PBT, PBT blends**

This brochure is based on many years of LANXESS experience and is intended principally as a guide. In certain cases it may be necessary to deviate from these recommendations.

Durethan® (polyamide) and Pocan® (polyester) polymers are two product lines that hold a high potential for growth and innovation. Our competitive production facilities and the intensive development work that we have conducted on products and applications have made us into a key supplier in many different markets.

The polymers business is also based on in-house production of the relevant feedstocks required. The production plants for cyclohexanol/cyclohexanone, caprolactam, adipic acid and glass fibers rank amongst the biggest of their kind.

MAIN SECTOR

Durethan® has a property profile that makes it ideal for applications in the automotive and electrical/electronics industries and in the construction sector.

Pocan® is used primarily in the electrical/electronics industry, although applications for this versatile material can also be found in the automotive industry, in medicine, and in the sports and leisure sectors.

2. SELECTION OF THE MACHINE AND PERIPHERALS

2.1 Determination of clamping force

General formula:

$$\text{Clamping force} \geq \text{mold opening force in kN} = \frac{\text{projected surface in cm}^2 \text{ x mean cavity pressure (opening pressure) in bar}}{100}$$

The actual clamping force required is determined first and foremost by the two variables included in the formula. Over and above this, the clamping force is influenced by other factors, such as the rigidity of the machine and the mold, the design of the molded part, the permitted breathing, the processing parameters and the molding compound itself.

- Projected surface = sum of all the surfaces subject to pressure projected onto the plane of the clamping platen
  Example: truncated cone-shaped disc:

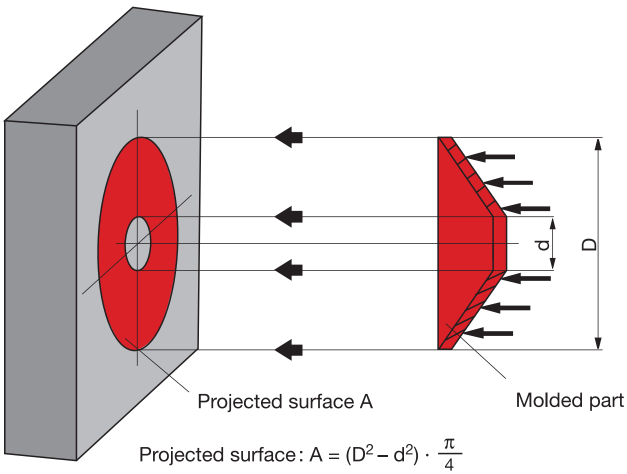


Fig. 1: Projected surface of a molded part (schematic)

Mean cavity pressure (opening pressure):

The pressure in the cavity is not uniform throughout the mold. The pressures close to the gate are much higher than those at a distance from the gate.

The mean cavity pressure is thus used when calculating the clamping force. This can be estimated on the basis of filling simulations. When designing molds for Durethan® and Pocan®, a maximum filling pressure (without the gate system) of 650 bar is frequently taken as a basis. When the cavity has been completely filled, a pressure of 650 bar acts at the gate and approximately 1 bar at the end of the flow path. In this example, the mean cavity pressure with a linear pressure profile would be 650 : 2 = 325 bar.

In practice, however, it is generally necessary to increase this value by a considerable margin in order to make allowance for the part geometry and possible pressure peaks prevailing during the switchover from injection pressure to holding pressure. In extreme cases, it may even be necessary to offset almost the entire injection pressure. Values of 250 to 700 bar are, however, frequently employed for the calculation.

Example:

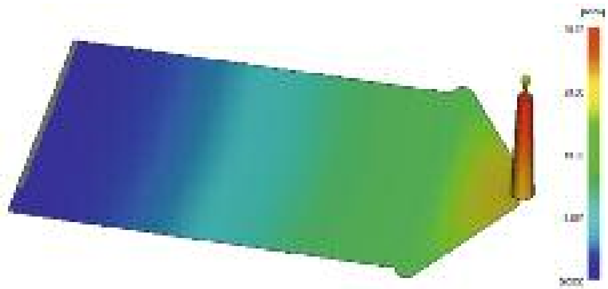


Fig. 2: Example of the pressure profile in a sheet mold

2.2 Screw geometry

Three-zone screws with an L : D ratio of between 18 : 1 and 22 : 1 and a flight depth ratio of 2 : 1 to 2.5 : 1 are suitable for use with LANXESS thermoplastics.

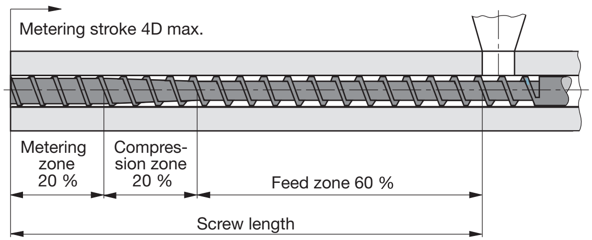


Fig. 3: Three-zone screw

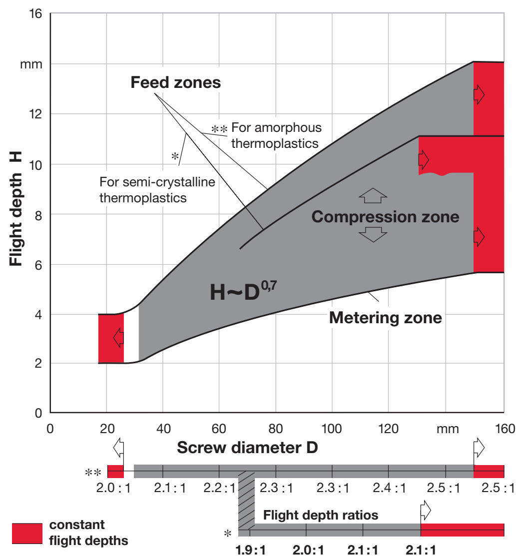


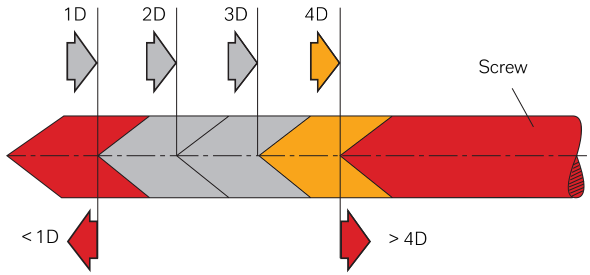
Fig. 4: Correlation between screw diameters, flight depths and flight depth ratios for standard three-zone screws

In the case of Durethan® and Pocan®, it is a good idea for screws with diameters in excess of 80 mm to have a reduced flight depth in the feed zone.

2.3 Determination of screw diameter, shot weight and metering stroke

It is best to coordinate the shot weight and the screw diameter in such a way that the metering stroke works out at between 1 x D and 3 x D (D = diameter).

If the machine size falls outside this range, the quality of the moldings can be impaired through molecular weight reductions due to an excessively long residence time, or through surface defects caused by air trapped in the screw.



1D to 3D optimum range
3D to 4D possible in exceptional cases
< 1D and > 4D not recommended

Fig. 5: Suitable and feasible metering strokes for injection molding screws

The following diagram shows the relationship between the shot weight and a suitable screw diameter:

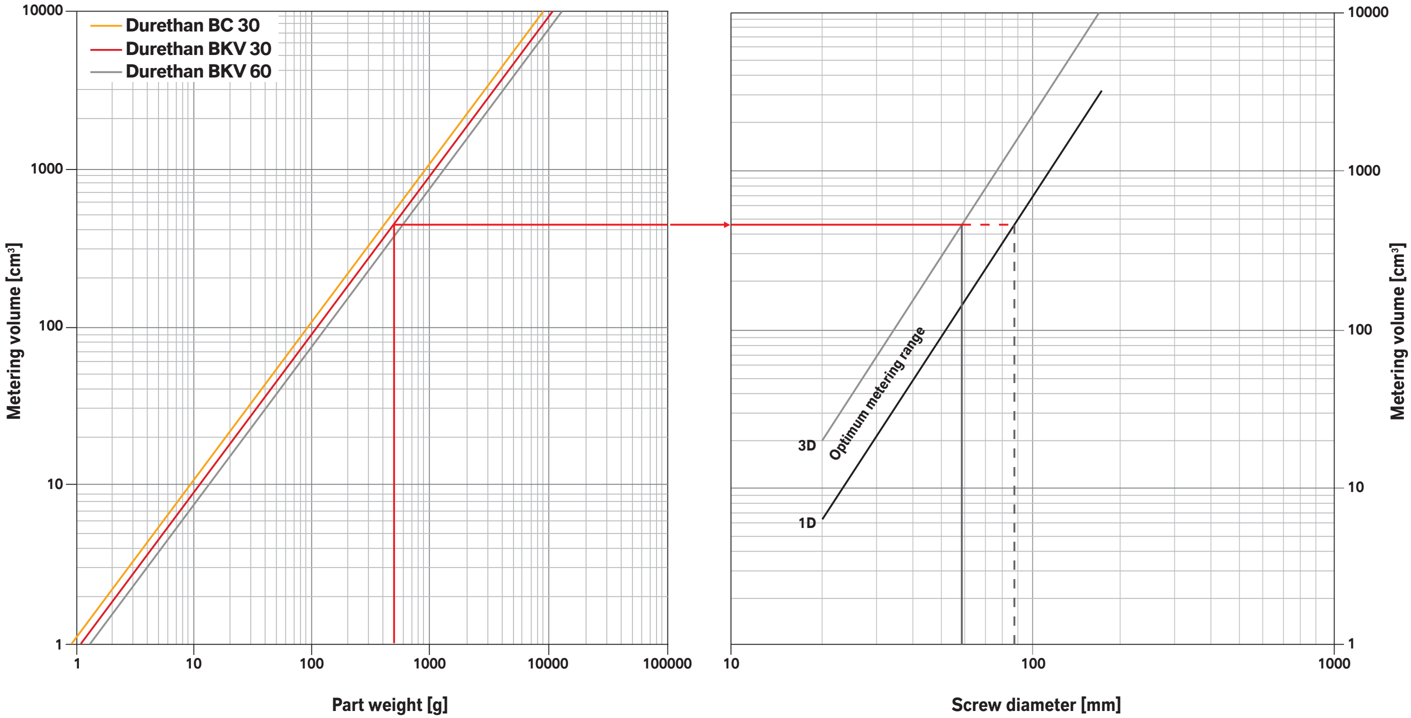


Fig. 6a: Correlation between screw diameter, metering volume and part weight for the injection molding of Durethan®

The nomogram in figures 6a & b shows the correlation between screw diameter, metering volume and part weight when thermoplastics are processed on injection molding machines.

This can be used both for establishing the screw diameter (machine size) for a known part weight and for estimating the minimum or maximum part weight for a given screw diameter. The nomogram is based on the use of a three-zone screw (L : D ratio 18 : 1 to 22 : 1) and an optimum metering stroke in the range 1D to 3D.

Taking as an example a part in PA 6 GF (in this case, Durethan® BKV 30) with a weight of 500 g, including the sprue, the nomogram shows that use should be made of a minimum screw diameter of 57 mm (using the maximum metering stroke of 3D) and a maximum screw diameter of 82 mm (using the minimum metering stroke of 1D). Conversely, for a screw diameter of 25 mm, for instance, and a non-reinforced PBT (in this case, Pocan® S 1506 – Fig. 6b), it is possible to read off a minimum part weight of approximately 12 g (with the minimum metering stroke of 1D) through to a maximum part weight of approximately 38 g (with the maximum metering stroke of 3D).

The metering volumes are determined by the melt density of the polymer. The higher the density, the smaller the screw diameter required.

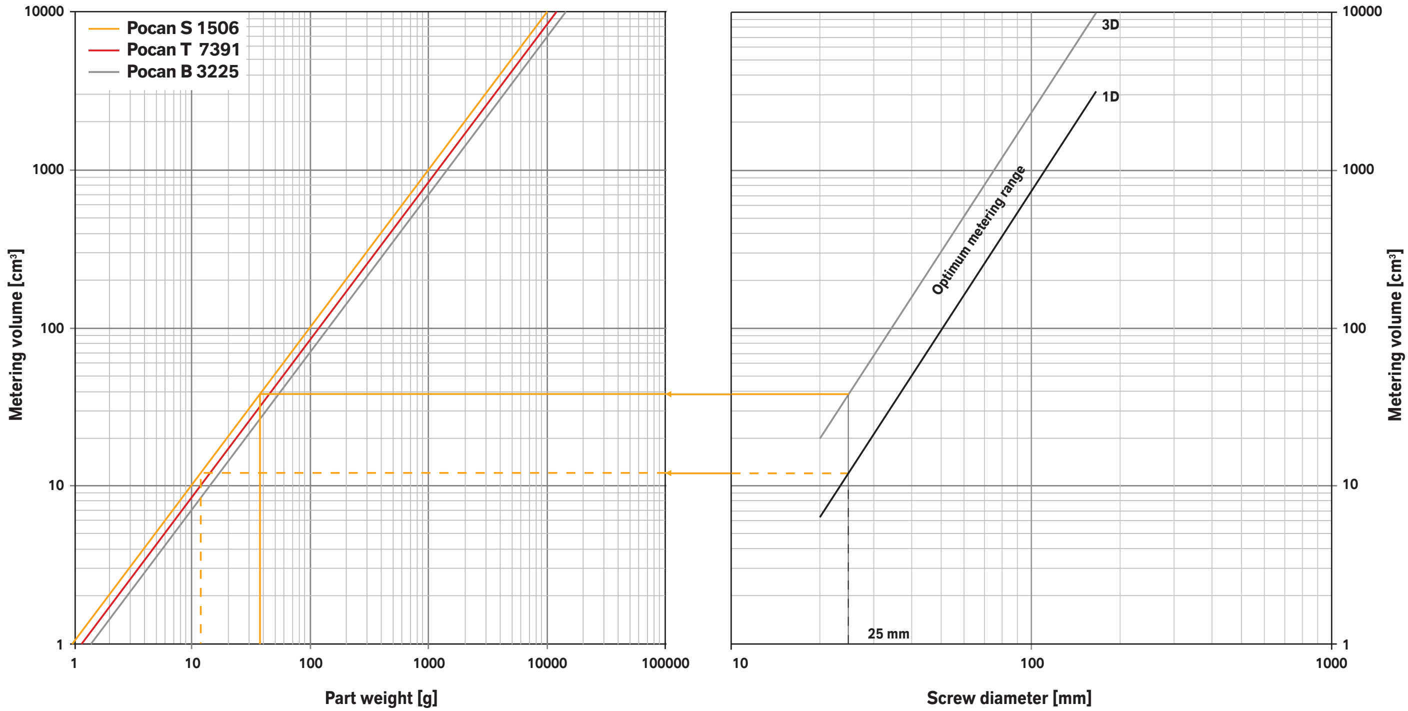


Fig. 6b: Correlation between screw diameter, metering volume and part weight for the injection molding of Pocan® (see preceding page for explanations)

2.4 Nozzles

Open nozzles should be used wherever possible. Shut-off nozzles can also be used for easy-flow materials, although, depending on their design, they can more readily lead to problems such as material degradation, specks and malfunctions (see also the points set out below).

- Spring-loaded needle systems lead to higher injection pressure requirements and to higher short-term material shear. Systems that are hydraulically or pneumatically driven on both sides do not suffer from these disadvantages, nor do mechanically controlled sliding shut-off nozzles.
- The success of all needle and sliding shut-off nozzle systems depends to a large extent on the melt channel being suitably designed in flow engineering terms (no dead spots or flow divisions).
- On all shut-off systems, movable actuating elements should be fitted with a certain amount of "play" so as to permit "melt lubrication" and an intentional, slight leakage flow to the outside.

On all nozzles, care should be taken to ensure a good fit between the diameter of the nozzle aperture and the diameter of the gate.

Guide values:
Nozzle aperture = gate diameter minus 0.5 to 1.0 mm

2.5 Protection against wear

As with all mechanical equipment, the plasticating unit is subject to wear as thermoplastics are processed. A basic distinction is drawn here between abrasion and corrosion. These can occur in isolation and also together.

Wear on components is frequently only detected at a late stage once malfunctions occur. In many cases, however, this wear will have started to affect the molded parts much earlier on, in the form of discoloration of the surface or similar defects. Sometimes these defects are located inside the molding and are not visible on the surface.

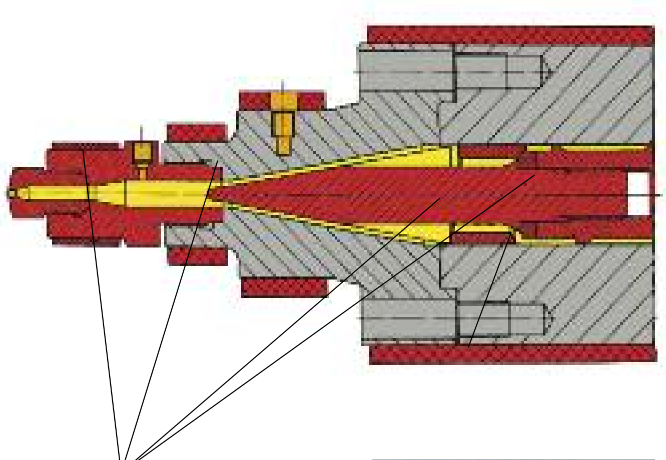
High costs are incurred not only through replacing worn, unserviceable machine components, such as screws, barrels and non-return valves, but also through rejected parts and the reduced availability of the machines due to stoppages and repair time.

When processing Durethan® and Pocan®, it is therefore advisable to invest in wear and corrosion-protected units. When selecting the grade of steel and the surface treatment method, knowing which of the two wear mechanisms predominates can be decisive. It is best to use abrasion-protected machines for glass fiber reinforced and mineral-filled plastics, while corrosion-protected machines are worthwhile especially for products with halogen-containing flame retardants.

2.6 Sealing faces: Nozzle, nozzle head and non-return valve

One frequent cause of wear problems is non-intact sealing faces inside the plasticating unit. Melt that penetrates the gaps in these non-intact sealing faces becomes damaged (dead spots, residence time and temperature) and is picked up again by the new melt flowing past it. The damaged melt can then lead to dark streaks, cloudiness or specks in the molded parts.

- When assembling the plasticating unit, bedding-in paste (applied as thinly as possible) should be used to ensure that the sealing faces are fully in contact with each other.
- Attention should be paid to the detailed instructions supplied by machine manufacturers on the correct assembly of the individual components, such as the barrel head and the nozzle.



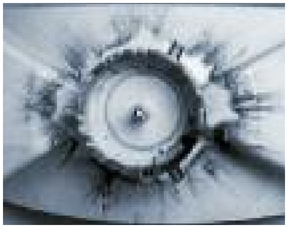


Fig. 7: Non-intact sealing face on the front end of the screw with degraded melt right up to the threaded hole

Fig. 8: Molding displaying pronounced discoloration on account of degraded melt

2.7 Temperature control of the mold

The mold temperature has a decisive influence on molded part quality and can particularly affect such properties as inherent stresses, warpage, dimensional tolerances, weight and surface finish. The cooling time is also determined to a large extent by the mold surface temperature.

It is only possible to comply with production specifications, and particularly with dimensional tolerances, if a defined mold temperature is maintained. As a rule, the heating/cooling equipment employed to this end can only ensure a constant mold temperature, at a specific level, with certain limitations. First of all, the cavity surface is heated up by 5 to 15 °C during the injection phase as it comes into contact with the melt.

By the time the next injection cycle commences, this temperature increase will have been offset once again through the removal of heat. With a steady-state cycle, therefore, a periodic temperature fluctuation will result (a "saw tooth" profile). During production start-up, however, the mold temperature will increase for a certain period of time, until a state of equilibrium has been achieved between the supply and the removal of heat. Superimposed on this will be the (at times quite considerable) control fluctuation of the temperature control unit and heat losses.

The mold temperature can thus deviate considerably, both upwards and downwards, from the setpoint value on the temperature control unit. It is therefore advisable to establish the actual mold temperatures on the basis of measurements and to correct the control system accordingly.

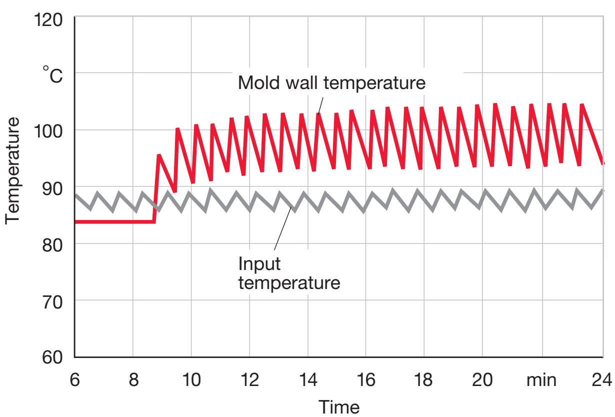


Fig. 9: Example of a temperature profile

The equilibrium temperature and the time taken for thermal equilibrium to be attained are a function of the heating/cooling medium throughput and the flow resistance. The flow resistance is determined by the number of heating/cooling channels and sharp changes of direction in the mold (more than one heating/cooling circuit connected up in a series arrangement).

In many cases, the pump on the temperature control unit does not supply sufficient pressure for the requisite throughput of heating/cooling medium to be achieved (10 to 15 l/min). In other cases, the maximum pressure level may be kept very low by a pressure-limiting valve.

This results in a "creeping flow" and hence in an insufficient exchange of heat in the mold. The temperature differential between the inflow into the heating/cooling unit and the outflow from it provides an indication as to whether the throughput is too low. This differential should be less than 4 °C.

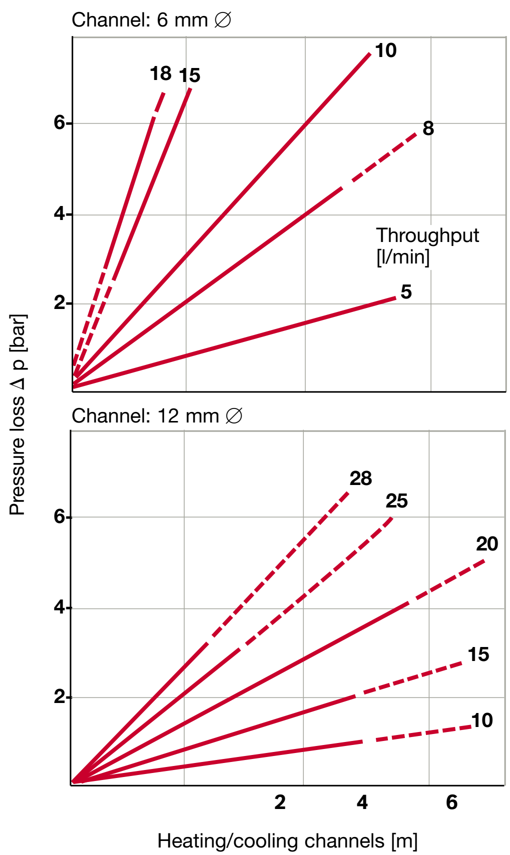


Fig. 10: Pressure losses in heating/cooling channels of different diameters

One essential condition for the rapid attainment and reliable control of the required mold temperature is a sufficient heating and cooling capacity in the temperature control units employed. The following diagram provides guide values for the heating capacity, as a function of mold size and mold temperature.

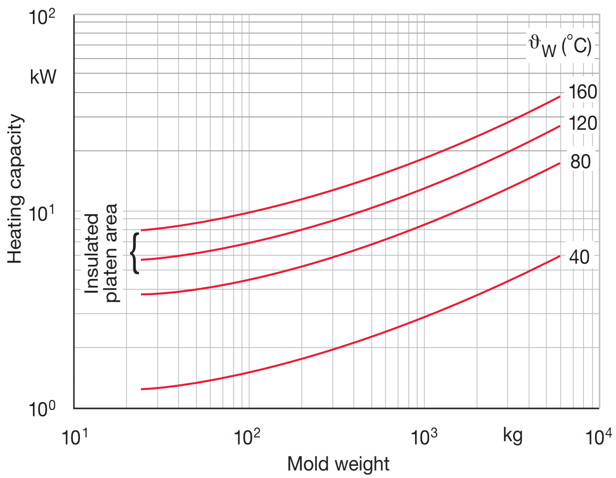


Fig. 11: Requisite heating capacity as a function of mold size for different temperatures

3. PROCESSING

3.1 Drying

Most polymer granules gradually absorb moisture from the air when stored. Even small quantities of moisture can cause problems during injection molding. In the case of polyamide 6, visible surface defects can develop, in the form of streaks (water vapor). With PBT, any water that is present is first used up, causing the polymer chains to split. This then gives optically flawless, but brittle, parts. Unless the granules have been stored in moisture-proof packaging, Durethan® and Pocan® must be dried prior to use.

With Durethan®, in particular, it is important not to overdo the drying, since this can cause discoloration and clearly reduced flowability (Fig. 12). We thus recommend not exceeding a drying temperature of 80 °C and using dry-air dryers. In our experience, fresh air and circulating dryers are not suitable for polyamides at 80 °C – in the extreme case, the granules can contain more moisture after drying than they did beforehand. For purposes of estimating the necessary drying time, it is useful to know the moisture content at the outset. This can be measured by Karl Fischer titration. Moisture balances can also provide valuable assistance in practice, despite their generally lower precision.

	Drying				
	Temperature	Time (hours)			Residual moisture
	°C	Circulating dryer	Fresh-air dryer	Dry-air dryer	%
Durethan®	80	not suitable		2 to 6[1)]	0.03 to 0.12
Pocan®	120	4 to 8	2 to 3	1 to 4	0 to 0.02

[1)] will depend on the initial moisture content

Table 1: Recommended drying conditions and moisture content for injection molding

The above data refers to packages stored at room temperature. It is also assumed that the drying equipment is in perfect working order and that the recommended drying temperature is observed.

Dried granules should be processed as rapidly as possible while they are still hot, and the machine hopper should be covered with a lid. Packages that have been opened should be re-sealed tightly and their contents used up as rapidly as possible. Failure to do this can greatly extend the required drying time, especially with Durethan®.

When processing granules taken from moisture-proof packaging, it is generally possible to dispense with drying. The packages must, however, be given sufficient time to warm up to ambient temperature before they are opened. Otherwise condensation will form, which will have to be removed by drying the granules.

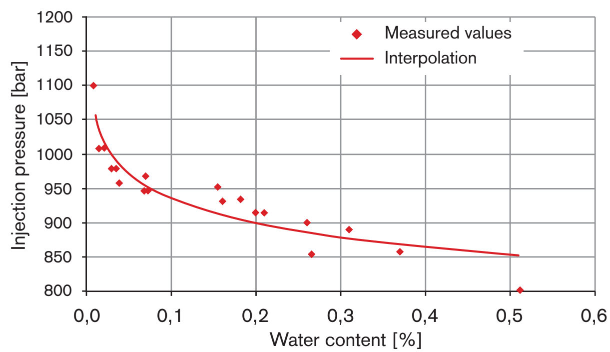


Fig. 12: Correlation between the injection pressure and the water content of the granules (taking the example of Durethan® BKV 30)

3.2 Mold and melt temperature, residence time

The figures given for mold and melt temperatures in the table below apply to general-purpose injection molding grades without flame retardants and can therefore serve only as a guide. A number of products require temperature settings that differ considerably from those specified. Please consult our data sheets and ask your contact at LANXESS.

The thermal stressing of the melt should be kept as low as possible in order to prevent undesired effects such as property changes in the plastic, the generation of decomposition products, clogged vents and mold corrosion. It is therefore important to avoid high melt temperatures and long residence times, due, for instance, to the use of a machine that is too big for the shot weight or to long cycle times.

In the event of prolonged interruptions to production, the screw should be moved forward for this same reason, and the barrel temperature reduced, or the heating switched off altogether.

With particularly temperature-sensitive products, it is a good idea to purge the barrel and, where appropriate, the hot runner too with a general-purpose product beforehand. After interruptions, the machine should always be purged with fresh granules.

	Mold temperature °C	Melt temperature °C
Durethan® PA 66 non-reinforced	80 to 100	275 to 295
Durethan® PA 66GF	80 to 120	280 to 300
Durethan® PA 6 non-reinforced	80 to 100	260 to 280
Durethan® PA 6 GF	80 to 120	270 to 290
Pocan® PBT; PBT-GF	80 to 100	250 to 270
Pocan® PET; PET-GF	100 to 130	260 to 280

Table 2: Recommended mold and melt temperatures

Even with correct processing, it is possible for volatile components and decomposition products to be given off. To preclude any risk to the health and well-being of the machine operatives, tolerance limits for the work environment must be ensured by the provision of efficient exhaust ventilation and fresh air at the workplace in accordance with the Safety Data Sheet.

3.3 Screw speed and back pressure

As the granules in the barrel are transported forwards through the rotation of the screw, they rub against the hot cylinder wall and melt. The screw speed must not be too high while this is taking place, since the melt will otherwise overheat, causing damage to the polymer. Flame retardant and other temperature-sensitive grades should be processed with the lowest possible peripheral screw speed (v_p) of between 0.05 and 0.2 m/s. In the case of general-purpose grades, a speed of between 0.05 and 0.3 m/s is recommended. Easy-flow EF and XF grades without flame retardants can frequently be processed at screw speeds in excess of 0.3 m/s in order to optimize the cycle time.

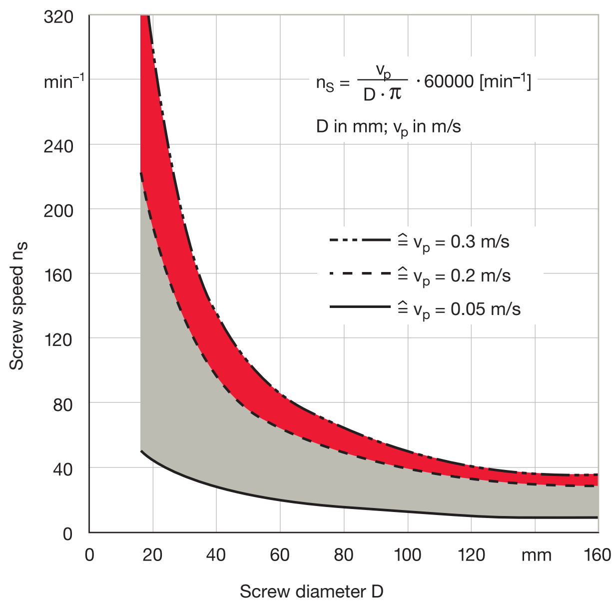


Fig. 13: Correlation between screw speed and screw diameter

The back pressures that will ensure even melting are normally in the order of 100 ± 50 bar (hydraulic pressure usually 5 to 15 bar).

Rules of thumb:
- To improve melt homogeneity: increase back pressure.
- To prevent uneven screw retraction (corkscrew effect): increase back pressure.
- Occasional interruption of melt transport: reduce back pressure.
- Metering time too long: reduce back pressure.

3.4 Injection and holding pressure phase

The injection and holding pressures, and also the injection speeds required, depend on the type of material being molded and the nature of the end product. The injection and holding pressure are set as hydraulic pressures. The latter must be high enough to achieve sufficient cavity pressure to enable the mold to be filled completely, without any sink marks. They can differ considerably for a given mold, depending on factors such as injection speed, melt temperature and nozzle geometry.

The injection speed is matched to the size and shape of the molded part and should generally be fast. The injection pressure should be high enough to ensure that the injection speed does not drop below the required setpoint value(s) during the entire injection process. If the injection speed drops towards the end of injection, this indicates that the injection pressure is too low or the set speed too high.

In order to avoid surface defects close to the gate (dull spots, cold slugs, delamination), it is a good idea to sharply reduce the speed at the start of the injection process (graduated injection). A constant flow-front speed can be achieved by implementing a velocity profile over the entire screw stroke (optimization of the filling process). In many cases, empirically-determined velocity profiles can help to remedy flow engineering problems (entrapped air, weld lines, bubbles, tear drops, streaks, diesel effect).

By reducing the speed directly prior to switchover, it is possible to level out the pressure profile and help prevent a backflow of melt.

The cavity pressure required for complete mold filling, the “filling pressure”, is a measure of the viscosity of the melt (providing that the filling time is kept constant). This can be used for process control purposes.

Another important factor is to switch over to holding pressure at the right moment in order to prevent overpacking of the mold.

Holding pressure serves to compensate for the volume shrinkage that occurs as the molded part cools in the mold. The level of holding pressure depends on the quality requirements of the molded part – e.g. dimensional stability, low stresses and surface properties (sink marks, reproduction) – and will generally be set as low as possible.

Holding pressure should be maintained until the gate system has “frozen” (in order to avoid any backflow of melt when the pressure is removed). The minimum holding pressure time (also known as gate open time) can be established through weight checks on the molded part (Fig. 15) or from the characteristics of the cavity pressure curve (Fig. 16).

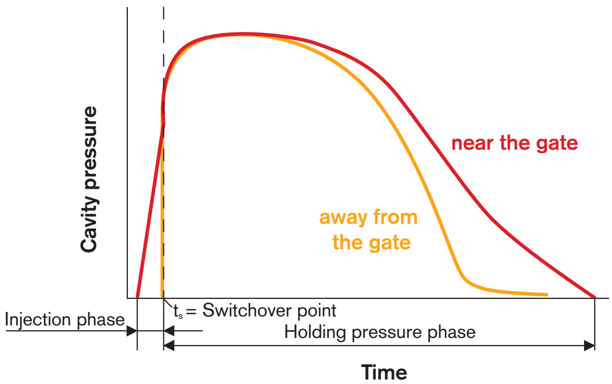


Fig. 14: Cavity pressure profile for semi-crystalline thermoplastics

The properties of the molded part are decisively influenced by process control.

The following are influenced during the injection phase:

- mechanical properties
- surface finish
- visibility of weld lines
- warpage
- completeness of cavity filling
- flash formation

The following are influenced during the holding pressure phase:

- weight
- dimensional stability
- shrinkage
- voids
- sink marks
- ejection characteristics
- weld line strength
- dimensional accuracy (warpage)

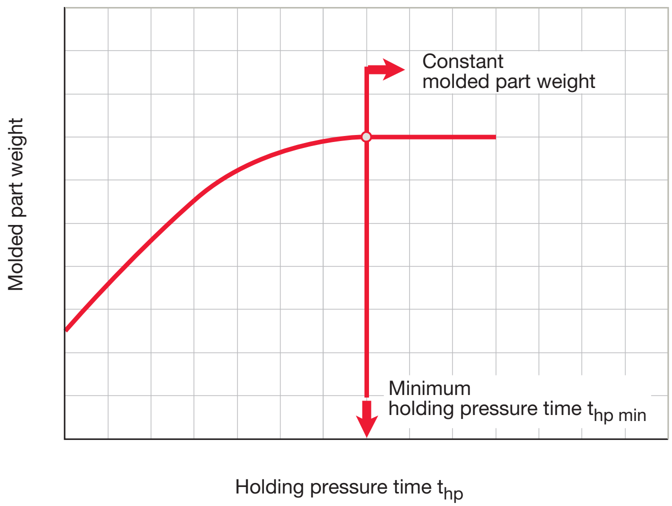


Fig. 15: Determining the holding pressure time from the increase in weight

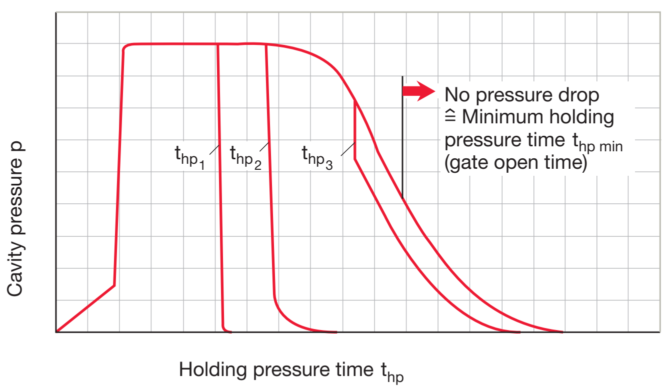


Fig. 16: Determining the holding pressure time from the cavity pressure curve

3.5 Cooling time

The following diagrams show the calculated cooling time of injection moldings (taking sheets as an example) as a function of

- material type
- wall thickness
- mold temperature (ϑ_W)
- melt temperature (ϑ_M).

The essential factors that influence cooling are wall thickness and mold temperature. Melt temperature has only a slight influence on cooling time.

NB: Cooling time is understood here to be the time from the initial application of holding pressure through to the point of demolding.

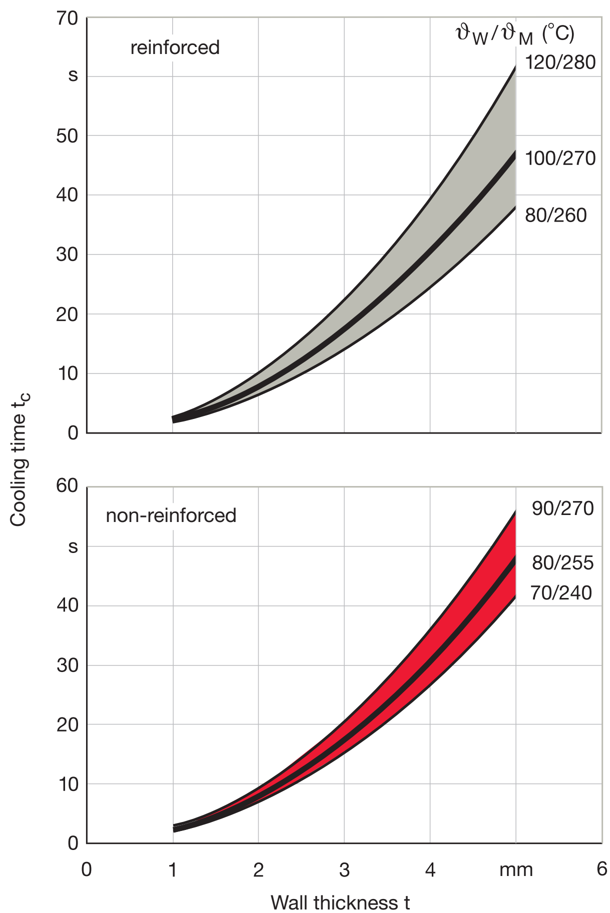


Fig. 17: Cooling time/wall thickness diagram for Durethan®

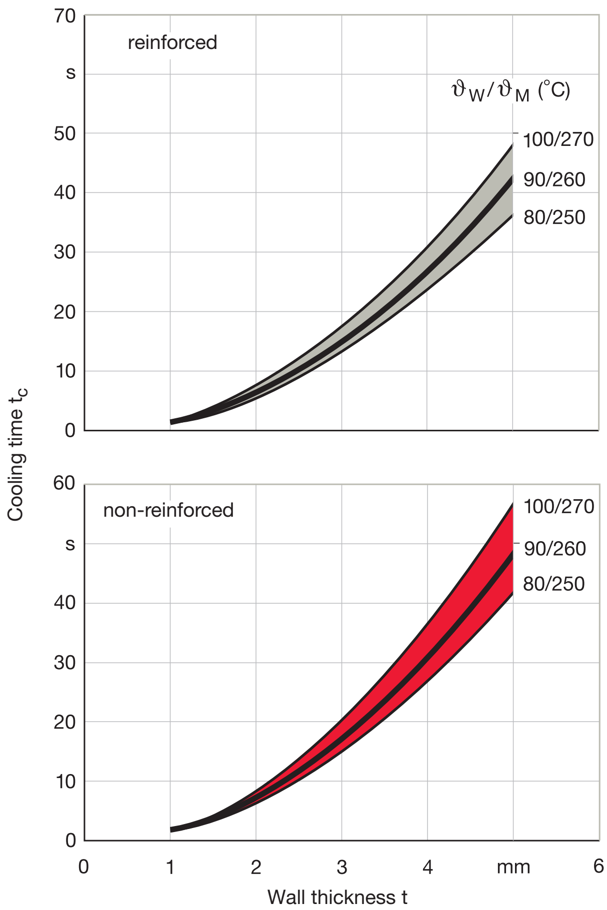


Fig. 18: Cooling time/wall thickness diagram for Pocan®

3.6 Cleaning the plasticating unit

To prevent any loss of time and material, it is best to switch from light colors to dark colors and from low viscosities to high ones when changing to a different material. The plasticating cylinder can be purged with appropriate high-viscosity molding compounds to clean it (PE, PP, PMMA, SAN, PS).

- In the case of severe encrustation (e.g. boundary layers adhering to the wall), pre-clean the unit with barrel cleaning agent. Additionally purge with high-viscosity PE or PP where necessary.

- Dismantle unit if necessary and clean components with a steel brush while still hot, then polish with a cloth and polishing paste (cf. the Safety Data Sheet!).
  Do not use sandpaper. Do not blast with glass or steel balls.

- As an alternative, the dismantled components can be cleaned in aluminum oxide vortex baths, oil baths or solvent baths (with ultrasound if required).
  It is important to note the Safety Data Sheets and the disposal regulations.

3.7 Processing reclaim

Molded parts in Durethan® (PA 6, PA 66, co-polyamides) and Pocan® (PBT, PET and PBT blends) can be ground and re-melted, observing the recommended drying and processing conditions. This may, however, cause damage to the polymer and the additives, which will have a detrimental effect on the properties of the finished parts. This effect can be reduced by mixing reclaim with virgin material of the same type. The permitted ratio must be checked for each application individually, with allowance also being made for external requirements, such as those prescribed by testing organizations for electrical appliances.

Points to note when processing:
- reject parts and sprues that are collected, ground and dried should be of a single sort
- contamination with oil, other plastics and dirt, etc. should be avoided
- fine components (dust) should be removed after shredding where possible
- the pellet size of the reclaim should correspond approximately to that of the virgin material
- uniform mixing of reclaim and virgin material is essential
- melt cake and moldings showing signs of overheating should not be used (thermal degradation)
- parts with streaks caused by moisture should be avoided if possible

We recommend that the amount of reclaim that can be added in each individual case be established through the appropriate tests (e.g. molecular weight reduction, mechanical properties). Assistance on this subject can also be obtained from your customer service representative at LANXESS.

3.8 Coloring with masterbatches

Like a large number of other thermoplastics, Durethan® and Pocan® can be colored using masterbatches.

Advantages:
- flexibility when coloring small series
- cost savings when making purchases and also for storage

Drawbacks:
- it may not be possible to compensate for fluctuations in the natural color of the polymer
- there may be color inhomogeneities in the finished parts
- product properties can be unfavorably affected
- no UL conformity
- liability in the event of complaints may be disputed
- metering units are required and possibly additional drying capacity

The melt viscosities of the masterbatch and the plastic to be colored should be as similar as possible. The material taken as a basis for the masterbatch should always be the polymer of the plastic that is to be colored. Problems can otherwise be encountered, such as segregation (streaks, delamination), insufficient adhesion of inks or sealing compounds, reduced impact strength, or undesirable long-term effects, such as a more pronounced tendency to yellowing in heat, or poor weathering resistance.

Poor homogenization can potentially be remedied through the use of static mixers. In this case, however, particular attention should be paid to the amount of time and material consumed by color changes and the potential influence on the product properties.

4. MEASURES FOR THE ELIMINATION OF MOLDING FAULTS

Contents	Page

Fault	Possible appearance	Possible causes	Suggested remedy
Impurities in compound	gray foreign particles which appear shiny, depending on angle of light	abrasion from feed pipes, containers and feed hoppers	pipes, containers and feed hoppers should not be in aluminum or tinplate but in steel or stainless steel pipe or sheet; pipes should be cleaned on the inside and be as straight as possible
	dark specks, discolored streaks	dust or dirt particles	keep dryer clean and regularly clean air filter, carefully close opened bags and containers
	colored streaks, surface layer near sprue comes adrift	presence of other plastics	separate different plastics, never dry different plastics together, clean plasticating unit, check subsequent batches for purity
Contaminated regrind	as for fresh compound (see above)	abraded material from pelletizer	check pelletiser regularly for abrasion and damage, and repair when necessary
		dust or dirt particles	store scrap away from dust, clean parts before pelletizing, discard parts containing moisture and thermally degraded parts
		other plastics regrind	always keep different types of regrind separate

Fault	Possible appearance	Possible causes	Suggested remedy
Moisture streaks	U-shaped, elongated streaks open towards flow direction; or, in a less pronounced version, in the form of small lines	residual moisture content of pellets too high	check dryer or drying process, measure pellet temperature, observe prescribed drying time
Silver streaks	elongated silvery streaks	overheating of melt due to too high melt temperature, too long residence time or too high screw speed; nozzle and runners too narrow	check melt temperature, use a more suitable screw diameter, reduce screw speed, widen nozzle and runner diameter
Streaks (entrapped air in compound or mold)	elongated streaks over a wide area, generally restricted to individual locations	injection speed too high, entrapped air due to incorrect metering, back pressure too low	reduce injection speed; increase back pressure within permitted limits, use optimum metering stroke (> 1D to 3D)

Fault	Possible appearance	Possible causes	Suggested remedy
	with transparent materials, bubbles may also be apparent as striations, black discoloration (diesel effect) at points where flows merge	entrapped air inside mold cavity	improve mold venting, especially near flow lines and near depressions (flanges, studs, lettering), correct flow front (wall thickness, gate position, flow leaders)
Burn streaks	brownish discoloration with streaking	melt temperature too high	check and reduce melt temperature, check temperature controls
		residence time too long	reduce cycle time, use a smaller plasticating unit
		unsuitable temperature profile in hot runner	check hot runner temperature, controls and thermocouples

Fault	Possible appearance	Possible causes	Suggested remedy
	occasional brownish discoloration with streaking	worn plasticating unit or dead spots near sealing face	check barrel, screw, non-return valve and sealing faces for wear and dead spots
		parts of the plasticating unit and hot runners impede flow	eliminate flow restrictions
		injection speed too high	reduce injection speed
Delamination	surface near sprue flakes off (especially with blends)	contamination through other, incompatible resins	clean plasticating unit, check subsequent material for purity
Gray streaks	gray or dark stripes, unevenly distributed	worn plasticating unit	replace whole unit or worn parts, use a plasticating unit with an abrasion and corrosion-resistant coating

Fault	Possible appearance	Possible causes	Suggested remedy
		dirty plasticating unit	clean plasticating unit
Cloudy appearance	extremely fine specks or metal particles in cloud formation	worn plasticating unit	see above
		dirty plasticating unit	clean plasticating unit
	cloud-like, dark discoloration	screw speed too high	reduce screw speed
Dark, generally blackish specks	less than 1 mm^2 to microscopic	worn plasticating unit	see above
	bigger than 1 mm^2	screw and barrel surface damaged and flaking off	clean plasticating unit, use unit with an abrasion and corrosion-resistant coating

Fault	Possible appearance	Possible causes	Suggested remedy
Dull spots	velvety spots near sprue, sharp edges, and changes in wall thickness	disturbed melt flow in gating system, at transitions from large to small-diameter runner and at bends (shear, tearing of already solidified outer skin)	optimize gate, avoid sharp edges, especially where gate joins mold cavity; round off transitions near runners and sudden wall thickness changes and polish them, inject in stages: slow – fast
Cold slug	cold melt particles entrapped in the surface	nozzle temperature too low, nozzle aperture too small	use band heater with higher capacity, fit nozzle with thermocouple and controller, increase nozzle aperture, reduce cooling of sprue bush, retract nozzle earlier from sprue bush

Fault	Possible Appearance	Possible causes	Suggested remedy
Voids and sink marks	round or elongated bubbles, visible only in transparent plastics, surface depressions	no compensation for volume contraction during the cooling phase	increase holding pressure time, increase holding pressure, reduce melt temperature and alter mold temperature (in the event of voids this must be increased, and in the event of sink marks, reduced), check melt cushion, increase nozzle aperture
		molded part does not have the right design for a plastic (e.g. wall thickness differences too great)	redesign part avoiding sudden changes in wall thickness and accumulations of material, adapt runners and gate cross-sections to part
Blisters	similar to voids but smaller diameter and more of them	moisture content of melt too high, also too high residual moisture content in granules	optimize drying, if necessary use a normal screw instead of a vented screw and pre-dry material; check dryer and drying process and use dry-air dryer if necessary
Jetting	surface with visible string of resin, near the gate	unfavorable gate location and size	prevent jetting by moving the gate elsewhere (inject against a wall), increase gate diameter

Fault	Possible Appearance	Possible causes	Suggested remedy
		injection speed too high	reduce injection speed or inject in stages: slow – fast
		melt temperature too low	increase melt temperature
Short moldings	incomplete filling of cavity, especially at end of flow path or near thin-walled areas	plastic does not have sufficiently good flow	increase melt and mold temperature
		injection speed too low	increase injection speed and/or injection pressure
		walls of part too thin	make walls thicker
		insufficient contact between nozzle and mold	increase nozzle contact pressure, check radii of nozzle and sprue bush, check centering
		diameter of gating system too small	enlarge gate and runner

Fault	Possible appearance	Possible causes	Suggested remedy
		mold venting inadequate	improve mold venting
Weld line strength insufficient	clearly visible notches along weld line	plastic does not have sufficiently good flow	increase melt and mold temperature, improve flow conditions by moving gate elsewhere if necessary
		injection speed too low	increase injection speed
		walls too thin	increase wall thickness
		mold venting inadequate	improve mold venting
Warped moldings	parts are not flat, are distorted, do not fit together	wall thickness differences too great, different flow speeds inside mold, glass fiber orientation	redesign part, change position of gate
		mold temperatures unsuitable	heat mold halves to different temperatures

Fault	Possible appearance	Possible causes	Suggested remedy
		unfavorable switchover point from injection to holding pressure	alter switchover point
Part sticks to mold	dull spots, finger-like or cloverleaf-shaped shiny hollows on surface (usually near sprue)	cavity wall temperature too high in certain places	reduce mold temperature
		part ejected too soon	increase cycle time
Part is not ejected or is deformed	part has jammed; ejector pins deform part or penetrate it	mold overloaded, too deep undercuts, cavity insufficiently polished near flanges, ribs and studs	reduce injection speed and holding pressure, eliminate undercuts, re-work cavity surfaces and polish in longitudinal direction
		vacuum is formed between part and mold during removal	improve mold venting

Fault	Possible appearance	Possible causes	Suggested remedy
		elastic deformation of mold and core displacement through injection pressure	increase stiffness of mold, support cores
		part ejected too soon	increase cycle time
Flash formation	Polymer melt penetrates mold gaps (e.g. parting line)	cavity pressure too high	reduce injection speed and holding pressure, bring forward switchover point from injection pressure to holding pressure
		mold parting surfaces have been damaged by overpacking	re-work mold near parting surfaces or contours
		clamping or locking force inadequate	increase clamping force or use machine with a higher clamping force

Fault	Possible appearance	Possible causes	Suggested remedy
Rough, matt part surfaces (with glass fiber reinforced thermoplastics)	rough, matt surfaces with flaky appearance; glass fibers visible	melt temperature too low	increase melt temperature
		mold too cold	increase mold temperature, equip mold with thermal insulation, use a more efficient heater
		injection speed too low	increase injection speed

Notes

4.1 Summary: Faults, causes, remedies

- ▲ increase, earlier
- ▼ reduce, later
- ▶ optimize (e.g. position)
- ◆ vary
- 1-7 Order for making changes

Fault	Melt temperature	Mold temperature	Injection speed	Injection speed (graduated)	Screw retraction	Screw speed	Switchover injection/ holding pressure	Holding pressure	Holding pressure time	Back pressure	Clamping force	Mold venting	Optimize gate	Cooling time	Check drying	
Streaks, stripes, specks, dots																
Streaks open towards the flow direction															▶	
Large-area silver streaks					▼ 2	▼ 3				▲ 1						Cushion
Streaks due to overheating, needle-like streaks	▼ 1		▼ 3			▼ 2							▲ 4			
Craters	▼ 1															Type, quantity of carbon black
White patches	▼ 2					▼ 3				▼ 4					▶ 1	(PA-GF)
Black or brown dots	Clean plasticating unit, soiled granules														▶	
Gray dots	Worn plasticating unit														▶	
Jetting				▼ 1									◆ 2			Deflector surface
Color																
Rings	▲ 1	▲ 2	▲ 3													
Homogeneous discoloration	▼ 1					▼ 2										Residence time
Black discoloration	Eliminate dead spots															
Darker color at weld line	▼ 2		◆ 1			▼ 3						▲ 4				
Partial color change			▼ 3			▼ 1				▲ 2			▲ 4			
Overheating marks (black)				▲ 1							▼ 3	▲ 2				
Cloudy appearance						▼ 1										Plasticating unit
Shiny appearance																
Dull spots				▼ 1									▲ 2			
Matt surface defects at hot runner elements		▼ 2												▲ 1		
Cold slug													▲			Ejector claw
Damage to grain on molded part	▲ 4			▲ 3			▲ 2	▼ 1								Optimize removal
Gloss differential on molded part surface		▶ 2	▲ 1													
Gloss level not achieved on polished surface		▶ 3	▲ 1					▲ 4	▲ 5			▲ 2			▶ 6	
Matt appearance not achieved on textured surface		▲ 2	▲ 1				▼ 3	▲ 4								
Frosting		▲ 2	▲ 1													
Removal behavior																
Sprue remains attached							▲ 1	▼ 2	▼ 3				▼ 5	▲ 4		
Scratches	▲ 3		▶ 4				▲ 1	▼ 2								Optimize removal
Noises as the mold opens							▲ 1	▼ 2								
Mold fails to open	▲ 3						▲ 2	▼ 1								
Stress-whitening				▲ 2				▼ 1								Optimize removal
Molding remains caught in cavity		◆ 6	▼ 1	▲ 3				▼ 2				▲ 4		◆ 5		
Part deformed during removal		▶ 4		▶ 1			▲ 3	▼ 2								Optimize ejector, surface
Ejector marks		▶ 4		▶ 1			▲ 3	▼ 2				▲ 5		◆ 6		Optimize ejector, surface
Fracture of part during removal		▶ 4		▶ 1			▲ 3	▼ 2				▲ 5		◆ 6	▶ 7	Optimize ejector, surface
Cracks, microscopic	▲ 1	▲ 2														Check media contact
Cracks, macroscopic			▼ 1				▲ 3	▼ 2								Optimize removal
Unevenness																
Sink marks, localized	▼ 3	▼ 4						▲ 1	▲ 2				▲ 5			Wall thickness/rib ratio
Large-area sink marks							▼ 1	▲ 2	▲ 3							
Notch along weld line	▲ 1	▲ 2	▲ 3									▲ 5	▶ 4			
Grooves	▲ 2	▲ 3	▲ 1													
Local, glossy, finger-shaped depressions							▼ 3	▲ 1	▲ 2					▲ 4		
Flakes	▲ 2	◆ 3		▼ 1												Especially for Mineral
Tear drops				◆ 1								▲ 2				
Delamination			▼ 2										▲ 1			Foreign material
Pockets	▲ 3					▼ 2				▲ 1						
Dimensions																
Flash	▼ 5		▼ 4				▲ 2	▼ 3			▲ 1					Sealing faces
Variations in size													▶		▶	
Variations in wall thickness							▲ 1	▼ 2			▲ 3					
Short moldings	▲ 1	▲ 2	▲ 3				▼ 4						▲ 5			
Weight variation											▲ 1				▶ 2	Non-return valve
Mechanical properties																
Mechanical problems with part, cracks	◆ 1	◆ 2						▼ 3							▶ 4	
Weld line strength insufficient	▲ 2	▲ 3	▲ 1									▲ 4	▶ 5			
Processing/plastication																
Voids								▲ 1	▲ 2				▲ 3			
Big bubbles					▼ 2					▲ 1						High cushion
Small bubbles															▶	
Cycle too long		▼ 4				▲ 2				▼ 1				▼ 3		
Unusual odor	▼ 1					▼ 2				▼ 3						Residence time
Warpage		◆ 3					▼ 1	▲ 2					▶ 4			GF orientation
Stringing		▶ 1			▲ 2								▼ 3			
Mold corrosion	▼ 2					▼ 3				▼ 4		▲ 1				Suitable steel types