ENGINEERING POLYMERS: THE 'TOP TEN' MOULDING PROBLEMS

By R. Wilkinson, E. A. Poppe, Karl Leidig, Karl Schirmer

The ten problems listed below occur often when moulding semi-crystalline engineering polymers such as POM, PA, PBT and PET. In this series of articles the authors describe simple ways to identify and avoid them.

Chapter 1. Moisture in the granules

1. Moisture in the granules
2. Feed system too small
3. Wrong gate position
4. Hold time too short
5. Wrong melt temperature
6. Wrong tool temperature
7. Poor surface finish
8. Problems with hot runners
9. Warpage
10. Mould deposit

	Symptoms when moulding	Visible symptoms in moulded parts	Influence on mechanical properties
PA	• Drooling • Bubble formation in the purge	• Splaying in direction of flow • Increased formation of flash	• Lower impact and tensile strength
PET PBT	• No noticeable symptoms	• N.B.: Surface streaks (splaying) are not visible	• Much lower impact and tensile strength
POM	• Bubbles may be formed in the purge • Some mould deposit may be formed	• There may be splaying	• None
TEEE	• No noticeable symptoms	• Slightly increased tendency to form flash	• Lower impact and tensile strength

How to recognise excess moisture content Source: DuPont

1.1

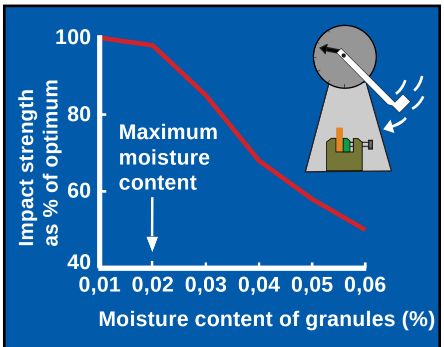
Many plastics absorb moisture from the atmosphere; how much they absorb depends on the type of resin. Moisture in the granules, even if it is only surface condensation, can cause problems in parts moulded with engineering polymers. Many kinds of undesirable effects can occur, including processing problems, poor surface on moulded parts, or loss of mechanical properties. It is seldom possible to establish whether there is moisture present by means of visual examination alone. The authors prepared this article to give moulders who process a broad range of plastics some useful guidance on how to handle those polymers that are sensitive to moisture.

PA	0,2 %	80°C	2 - 4 h	Needed only if resin has been exposed to atmosphere
PBT	0,05 %	120°C	3 - 4 h	Always needs drying (dehumidified-air dryer)
PET	0,02 %	130°C	3 - 4 h	Always needs drying (dehumidified-air dryer)
TEEE	0,1 %	80°C - 110°C	2 - 4 h	Drying temperature depends on hardness
POM	0,05 %	80°C	1 h	Only if you suspect surface condensation

Recommendations for maximum moisture content of the granules, drying temperatures and drying times

Source: DuPont

1.2



Influence of moisture content of the granules on the impact strength of glass-filled GF30 PET

Source: DuPont

1.3

Drying Plastic Materials

Most engineering polymers require the moisture in the granules to be below a certain maximum level for processing. The need for drying depends mainly on how sensitive the raw material is to water. Naturally, the moisture content of the material as delivered, the type of packaging and the period of storage are also important criteria. For example, polyamide is generally packed in bags with a barrier layer of aluminium, so that it can be used straight out of the bag. However, most processors of PA prefer to dry the resin in any case, even though drying is not necessary if the material is used within one hour.

PET and PBT, on the other hand, are far more critical where moisture is concerned and must always be dried to ensure that impact strength of the moulded parts is not affected. Another factor is that these resins pick up moisture very rapidly after drying, so that moulders should exercise special care when handling open containers of PET and PBT, when they are in transport or conveyor systems, as well as regarding their dwell time in the hopper. Thus, in unfavourable climatic circumstances PET can absorb enough moisture in 10 minutes to exceed the maximum permitted moisture content for moulding of 0,02 per cent. Drying regrind and fully saturated granules (e.g. in the case of containers which were left standing around open) requires special care. In these cases the recommended drying times are usually not enough. Fully saturated polyamide may need more than 12 hours to dry. The yellowing associated with such treatment is practically unavoidable. The following guidelines should therefore be followed:

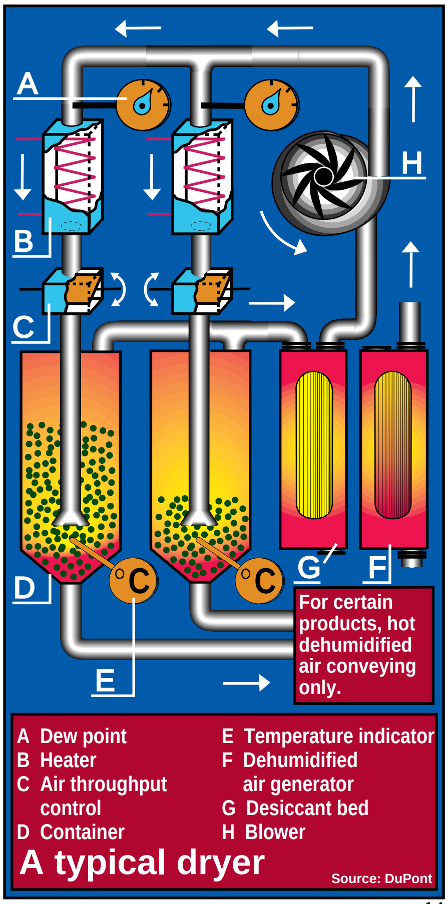
• Always store sprues and regrind in closed containers.
• Close containers or bags that have been partially used.
• Keep a lid on the hopper.

How to Dry

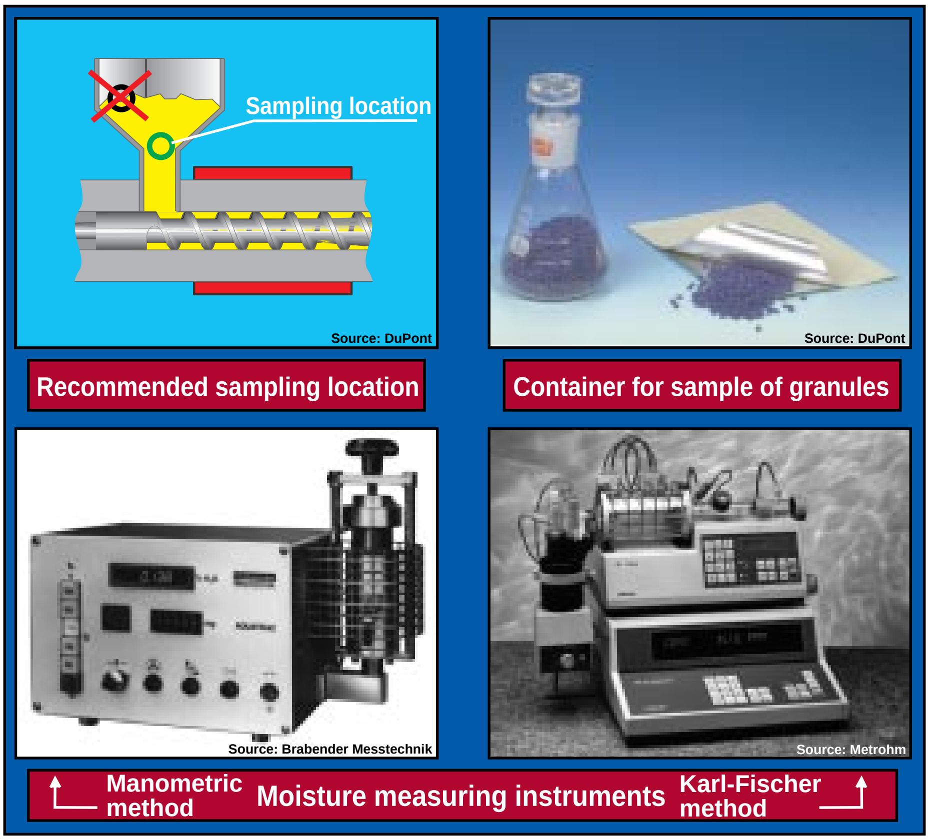
It is important to follow correct drying procedures if you want good quality mouldings. Simple hot air driers of various types are not adequate for drying polyesters, for example, but dehumidified-air drier systems are acceptable. Only these can provide the necessary constant and adequate drying, whatever the ambient climatic conditions may be. Apart from keeping the correct drying temperature, it is important to ensure that the dew point of the drying air remains lower than ≤–20° C. When operating multiple-container installations with different filling heights and bulk density, it is also important to ensure that the air throughput in each container is sufficient.

Measuring Moisture Content

Moisture in the granules can be measured with commercially available measuring instruments, e.g. with the manometric or the Karl-Fischer method. To eliminate sources of error, the sample should be taken from well down in the hopper, and should be sealed in an appropriate container. Special heat-sealable sachets coated with PE and aluminium are suitable, as well as laboratory-type glass containers that can be hermetically sealed.


A
B
C
D
E
F
G
H
°C
°C
For certain products, hot dehumidified air conveying only.
A Dew point
B Heater
C Air throughput control
D Container
E Temperature indicator
F Dehumidified air generator
G Desiccant bed
H Blower
A typical dryer
Source: DuPont

1.4


Sampling location
Source: DuPont
Source: DuPont
Recommended sampling location
Container for sample of granules
Source: Brabender Messtechnik
Source: Metrohm
Manometric method
Moisture measuring instruments
Karl-Fischer method

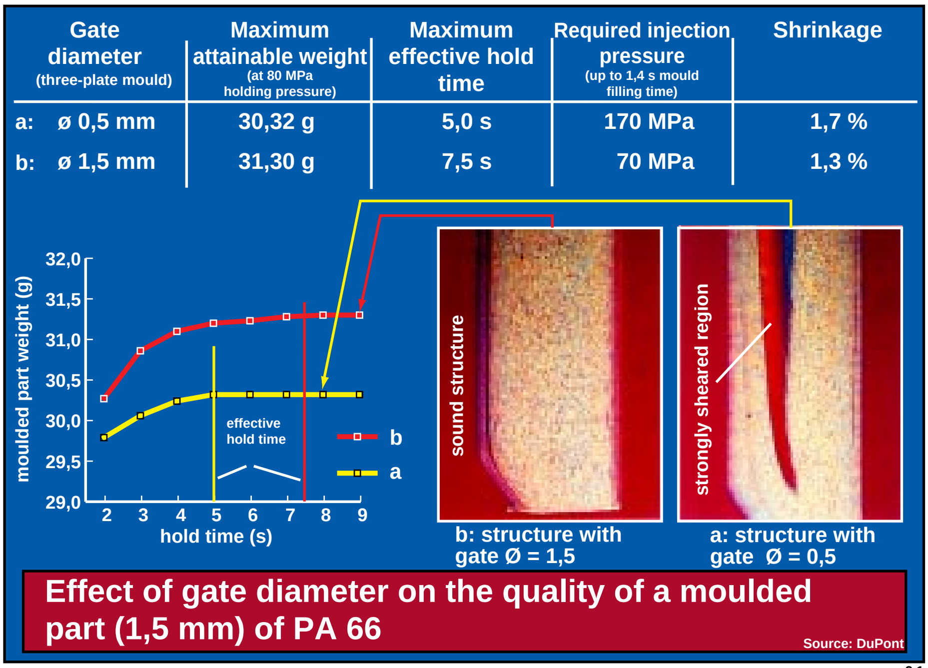
1.5

ENGINEERING POLYMERS: THE ‘TOP TEN’ MOULDING PROBLEMS

By R. Wilkinson, E. A. Poppe, Karl Leidig, Karl Schirmer

Chapter 2. Feed system too small

1. Moisture in the granules
2. Feed system too small
3. Wrong gate position
4. Hold time too short
5. Wrong melt temperature
6. Wrong tool temperature
7. Poor surface finish
8. Problems with hot runners
9. Warpage
10. Mould deposit

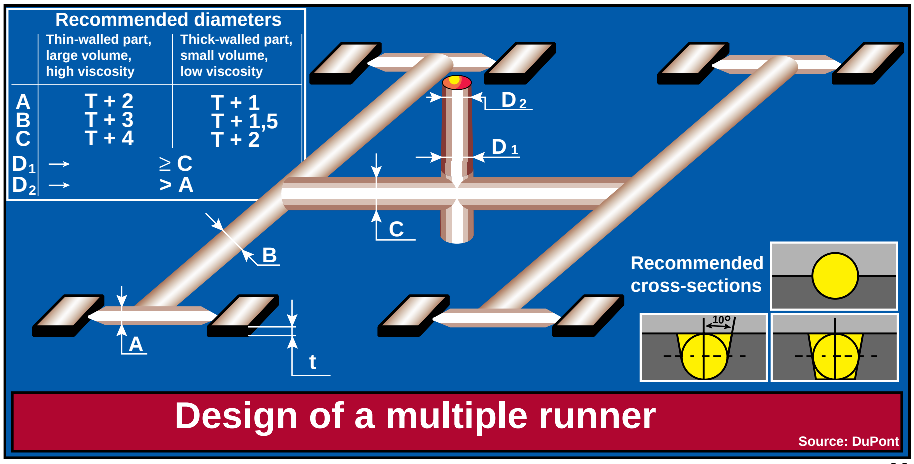


Gate diameter (three-plate mould)	Maximum attainable weight (at 80 MPa holding pressure)	Maximum effective hold time	Required injection pressure (up to 1,4 s mould filling time)	Shrinkage
a: ø 0,5 mm	30,32 g	5,0 s	170 MPa	1,7 %
b: ø 1,5 mm	31,30 g	7,5 s	70 MPa	1,3 %

Effect of gate diameter on the quality of a moulded part (1,5 mm) of PA 66

Source: DuPont

2.1



Design of a multiple runner

2.2

Parts made of engineering polymers nowadays are designed with the help of complex methods such as computer-aided design, finite element analysis and mould-flow calculations. Though they are unquestionably useful, they sometimes fail to take enough account of the importance of the correct design of the feed system. This article considers the basic elements of correct feed system design for semi-crystalline polymers. But these elements need to be applied in combination with a correctly positioned gate and the right hold time. These subjects will be dealt with in the following chapters of this series.

A Distinguishing Feature of Semi-Crystalline Resins

Semi-crystalline thermoplastics undergo volume shrinkage during the transition from the molten to the solid (crystalline) state. This shrinkage, which may be as much as 14 per cent, depending on the type of resin, has to be compensated during hold time by the supply of additional melt into the mould cavity. That can only be done if the gate cross-section is adequate to ensure the presence of a fluid centre during the holding phase.

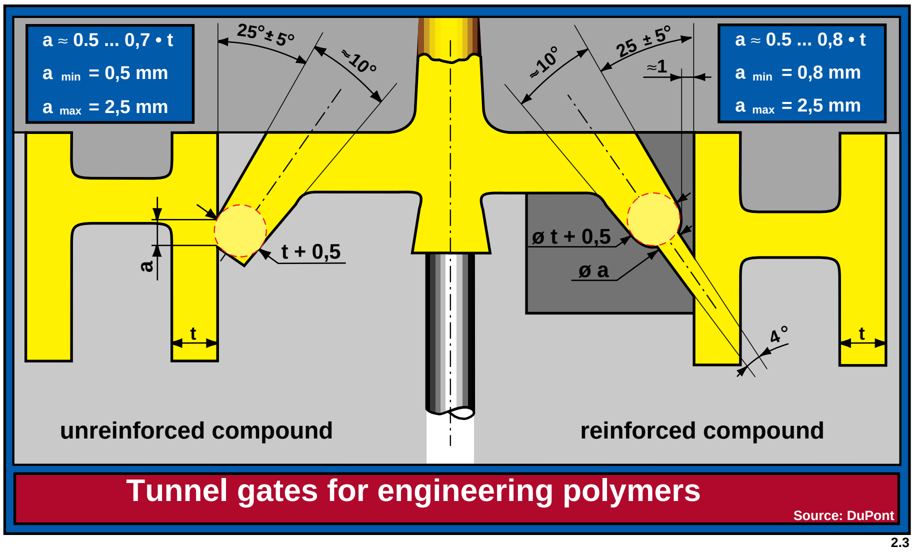
Recognising the Results

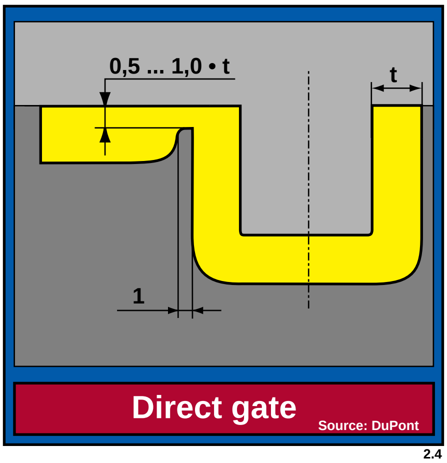
If the gating system is too narrow (see example), the holding pressure cannot remain effective beyond the desired holding pressure time. In that case, volume shrinkage cannot be adequately compensated, resulting in the formation of voids and sink marks, (especially in the case of unreinforced compounds), as well as pinholes (if the compounds are reinforced). These symptoms can be observed under the microscope. The dimensional stability of the resultant mouldings will also vary considerably, and there will be excessive shrinkage and an increased tendency to warp.

Voids and pinholes adversely affect mechanical properties since they act as notches and drastically reduce elongation at break and impact strength. In the case of fibre-reinforced compounds, the fibres will be damaged and become shorter if the gate is too narrow; this, in turn, will further weaken the moulding.

High injection pressures and long mould filling times can be a further indication that the gates are too narrow. This can be recognised, for example, by the fact that different injection rate settings have little effect on the actual mould filling time.

If the gate is too narrow, this can also cause surface defects. Excessive shear can result in additives such as impact modifiers, pigments, flame retardants and fibres separating. Gates that are too small will also tend to cause jetting, resulting in streaks, dull spots and a ‘marbled’ effect, and the formation of a kind of halo near the gate. There is also an increasing tendency for mould deposits to form.





Design of the feed system

In designing the feed system, the first point to be considered is the wall thickness (t) of the moulded part (see diagram). Nowhere should the diameter of the runner be less than the wall thickness of the injection moulding. Starting out from the gate, the runner diameter at each branch point can be widened so that an almost constant shear rate is maintained.

To prevent the inevitable cold slug reaching the moulding from the injection nozzle, the gate should always be extended so that the cold slug can be intercepted. This extension should have roughly the same diameter as the gate to ensure that the cold slug really is retained.

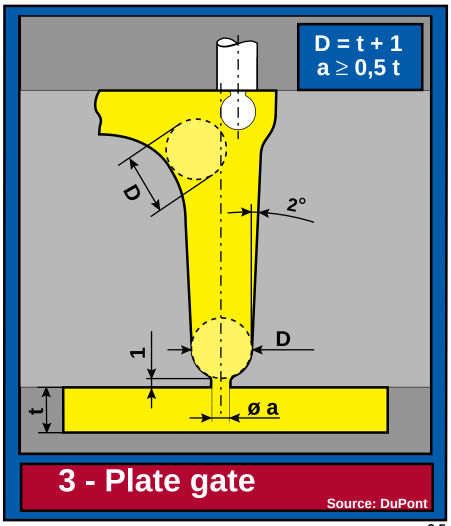
When moulding partially crystalline, unreinforced polymers, the minimum gate thickness should be 50 per cent of the wall thickness of the moulded part. This would also be adequate for reinforced compounds. To minimise the risk of damage to the fibres and also bearing in mind the higher viscosity of these compounds, the gate thickness should be up to 75 per cent of the wall thickness of the moulded part.

Gate length is especially crucial. This should be ≤1 mm to prevent premature solidification of the sprue. The mould will heat up near the gate, so that the holding pressure is at its most effective.

To summarise the basic rules:

• always provide a means of intercepting the cold slug;
• make the runner diameter bigger than the moulded part wall thickness
• gate thickness should be at least 50% of the moulded part wall thickness.

These principles take only the crystallising behaviour of engineering polymers into account. If one wants to estimate mould filling behaviour, data about flow lengths of the polymer can be used and, if needed, flow calculations must be carried out. There are probably certain applications where, for various reasons, gate design does not follow these recommendations. Here, one will generally have to compromise between quality and cost-effectiveness.

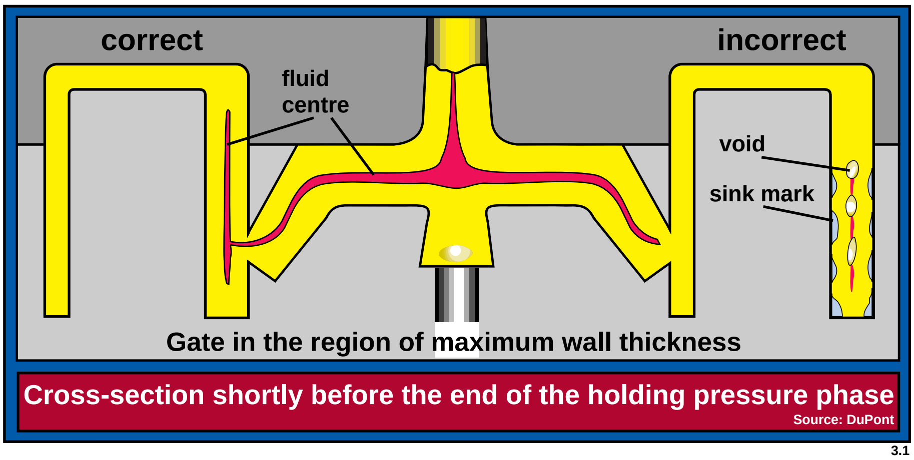


ENGINEERING POLYMERS: THE 'TOP TEN' MOULDING PROBLEMS

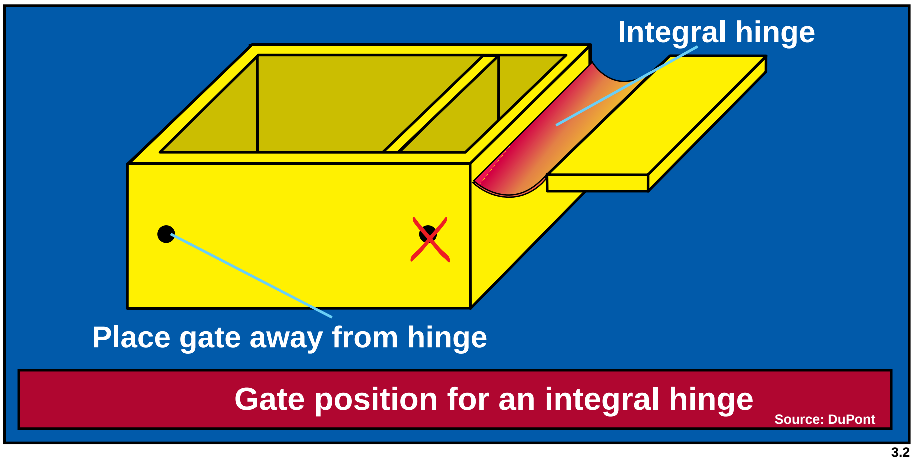
By R. Wilkinson, Roger Wilkinson, E. A. Poppe, Karl Leidig, Karl Schirmer

Chapter 3: Wrong Gate Position

1. Moisture in the granules
2. Feed system too small
3. Wrong gate position
4. Hold time too short
5. Wrong melt temperature
6. Wrong tool temperature
7. Poor surface finish
8. Problems with hot runners
9. Warpage
10. Mould deposit



The position of the gate is decisive for the flow front profile and the effectiveness of the holding pressure and, as a result, for the strength and other properties of the moulded part.
Since the position of the gate is usually specified by designers and mould makers, this article has been written especially with these persons in mind. Nevertheless, injection moulders should be also involved from the planning stage, to prevent predictable problems.



Possible negative consequences of poor gate position

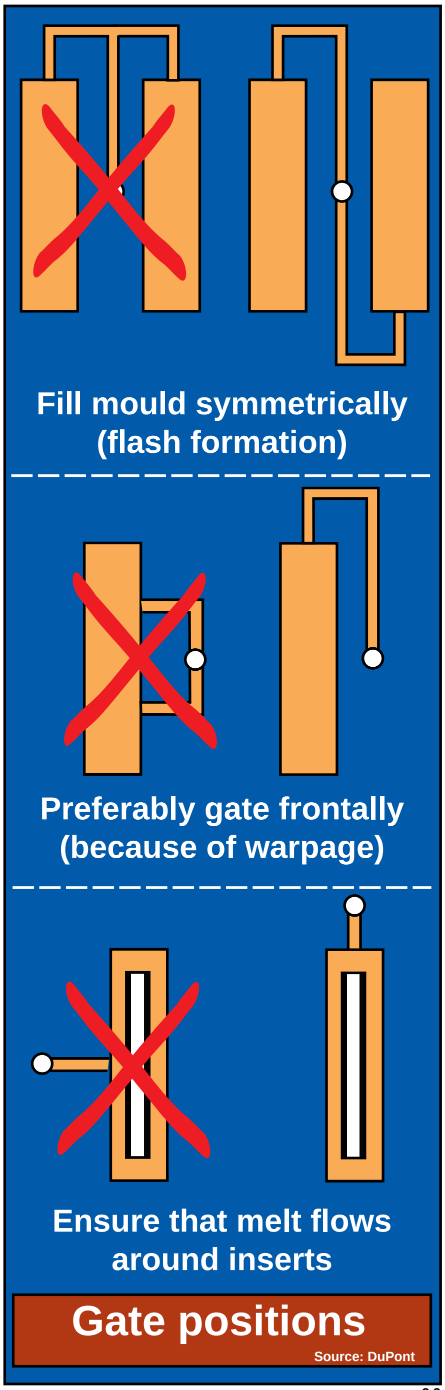
The properties of an otherwise correctly designed part made from a semi-crystalline engineering polymer can be ruined if the gate is not in the right position. This will be evident by the following symptoms, which apply to reinforced as well as unreinforced types of resin: weld lines and entrapped air, caused by the flow front profile, can influence the part's surface finish and, especially in the case of fibre-reinforced materials, its mechanical properties. Modifying processing conditions has no influence on these considerations.
Sink marks and voids are formed in the thick-walled part of a moulding if the gate is located in a thinner part of the moulding. Since the material crystallises sooner in the thin-walled section (see diagram), the thick-walled section, which requires a longer holding pressure time, can no longer be supplied with melt. Besides optical and mechanical problems, there will be increased shrinkage in that area, which can cause warping even in the case of unreinforced grades.
If the gates are too few and incorrectly positioned, flow distances can be too long and injection fill pressures too high. If the available mould locking force is insufficient or a polymer is being used which has low viscosity and crystallises too slowly, this can result in increased flash formation. Furthermore, the processing 'window' is greatly limited, so that it is no longer possible to fine-adjust tolerances via moulding conditions.

Recommendations for optimum gate position

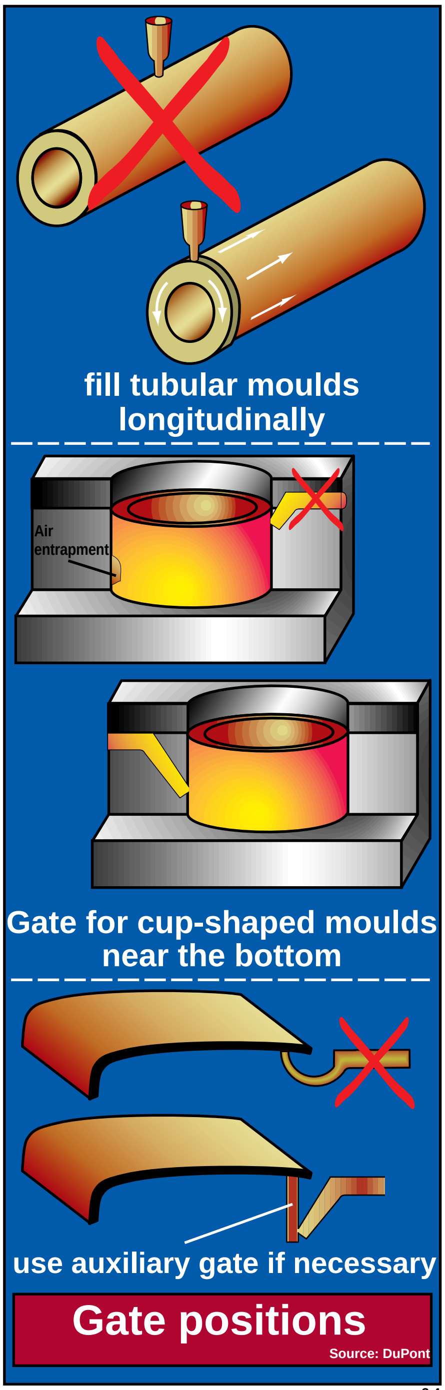
- Always try to gate into the region with the greatest wall thickness.
- Gates should never be near highly stressed areas.
- Long parts should be gated longitudinally instead of transversely or centrally, if at all possible, especially in the case of reinforced moulding compounds.
- If there are two or more mould cavities, the parts should be arranged and gated symmetrically in relation to the sprue.
- Axially symmetrical parts such as gear-wheels, discs, impellers, etc., should preferably be centrally gated using a diaphragm gate, or by resorting to multiple gating with a three-plate mould, in order to achieve good true running properties.
- Parts which include integral hinges should be gated so that the weld line will be away from the hinge. Flow stoppages near the hinges should be avoided aat all costs.
- Cup-shaped parts (e.g. small housings, capacitor cups, etc.) should be gated near the base, so as to prevent air entrapment.
- In the case of tubular parts, the melt should first be made to fill the annular circumference at one end and then fill the length of the tube itself. This will prevent an asymmetrical flow front profile.
- When insert-moulding around core pins, melt-out cores and other metal inserts, the molten resin should be able to flow round the insert in a circle, so as to keep misalignment of the insert to a minimum.
- Exposed surfaces which have to be free from visual defects such as gate marks can be gated from the underside, using a tunnel gate feeding onto an ejector pin.

- The gate should be positioned so that even brief flow front stoppages (complex parts, multi-cavity moulds with different shapes, etc.) during filling are prevented as far as possible.

These recommendations obviously cannot cover the entire range of possible applications. Compromises will always have to be made, depending on the complexity of a particular moulding. The recommendations we have discussed should nevertheless be taken into account during the planning stage, as far as possible. Simulated mould filling trials can be an invaluable aid in such situations



3.3



3.4

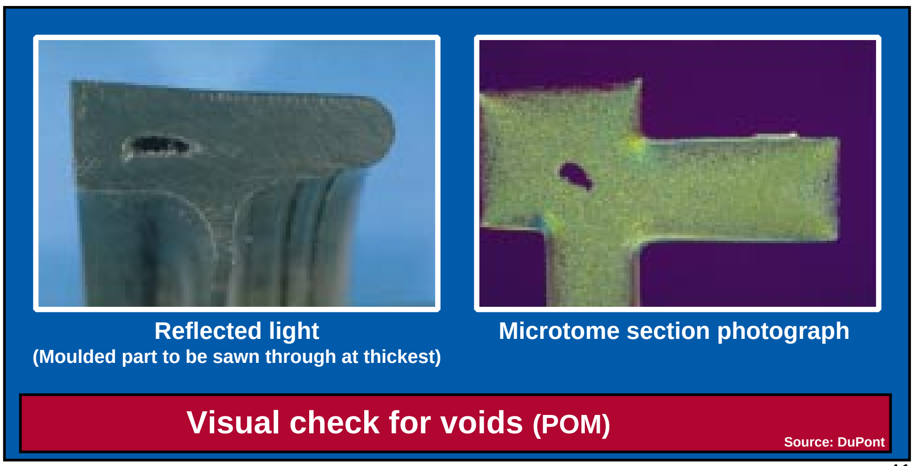
ENGINEERING POLYMERS: THE 'TOP TEN' MOULDING PROBLEMS

By R. Wilkinson, Roger Wilkinson, E. A. Poppe, Karl Leidig, Karl Schirmer

Chapter 4: Hold Time Too Short

1. Moisture in the granules
2. Feed system too small
3. Wrong gate position
4. Hold time too short
5. Wrong melt temperature
6. Wrong tool temperature
7. Poor surface finish
8. Problems with hot runners
9. Warpage
10. Mould deposit

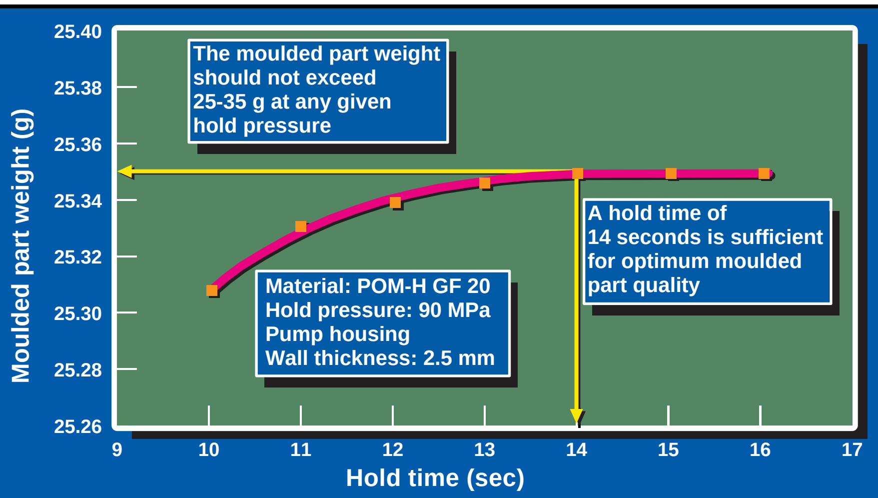
In practice many injection moulders, working from their experience of amorphous polymers, tend to use shorter hold pressure times and longer cooling times. Unfortunately, this approach also tends to be used for semi-crystalline polymers such as POM (acetal), PA (nylon), PBT and PET (polyesters). This article discusses the most important points to help machine setters chose the most suitable hold pressure time.



4.1

What exactly happens during the hold pressure phase?

Once the mould cavity has been filled, the polymer molecules start to crystallise, i.e. the molecule chains become aligned with respect to each other, resulting in higher packing density. This process starts in the outer zone and ends in the centre of the wall (see diagrams). The volume shrinkage caused by this can be as much as 14 %, as in the case of POM, and has to be made up again by further amounts of melt which are injected into the mould cavity during the hold pressure phase. If the hold pressure time is too short, it causes small voids to be formed (microporosity), which can have an adverse affect on moulded part properties in many ways.



The correct hold time is easy to determine on the injection moulding machine. Several different hold times are set, 1-2 seconds apart, depending on the required resolution, and the resultant mouldings weighed on a laboratory balance after removing the sprue. The weights are then plotted against the hold pressure times; this enables the optimum hold time to be easily determined. This will be in the region where there is no longer any change in the weight of the moulded part.

Experimental determination of effective hold time Source: DuPont

4.2

Material	Crystallisation time per mm wall thickness
POM - H	7,5 - 8,5 s/mm
PA 66	3,5 - 4,5 s/mm
PA 66 (impact modified)	3,0 - 4,0 s/mm
PA 66 GF 30	2,5 - 3,5 s/mm
PET GF 30	3,0 - 4,0 s/mm
PBT	3,5 - 4,5 s/mm
PBT GF 30	2,5 - 3,5 s/mm

Crystallisation rate for a wall thickness of 3 mm Source: DuPont

4.3

How to find out whether the hold time is too short

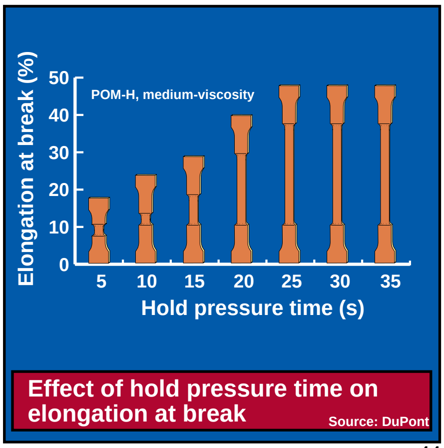
Parts made in this way often show excessive shrinkage, warpage, sink marks, voids and, in some cases, enormous loss of mechanical properties. In addition, there may be considerable dimensional variations. In some cases a misguided attempt is made to compensate for these shortcomings by increasing the cooling time. This results in unnecessarily long cycle times.

One way of recognising the effects of inadequate hold pressure times, for unreinforced moulding compounds, consists of cutting through the moulded part at a point where there is maximum wall thickness. The polished cut surface can then be checked for voids and pinholes. A magnifying glass or reflected-light microscope are sufficient to form a first opinion. A more elaborate method consists of preparing microtome sections (see diagram). In these, even the finest defects can be made visible with a transmitted-light microscope.

With reinforced moulding compounds, defects can be easily detected on a fracture surface where there is maximum wall thickness. If the hold time is too short, there will be a foam-like structure in the fracture region and an enlarged fracture photomicrograph will show exposed fibres which are not embedded in polymer. Another method consists of preparing a photomicrograph of a polished section, in which pinholes can be detected with a microscope.

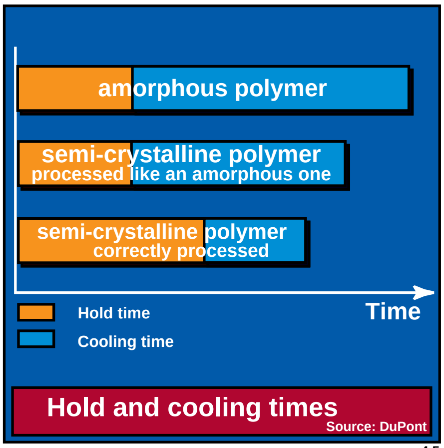
The effective hold time can be determined on the injection moulding machine by weighing a number of mouldings (see description). This is the best way of determining the hold pressure time for a given moulding under practical conditions.

A guideline for the optimum hold time can also be obtained by using actual figures as comparison (see table). These apply only for a given wall thickness and cannot be applied to other factors such as temperature, nucleating additives and pigments, mould filling time, etc. For thinner walls, the figures will be lower, for thicker ones higher.



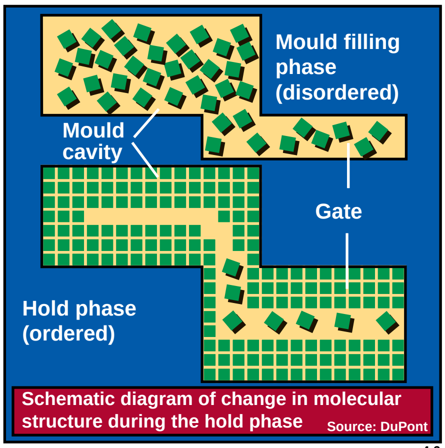
Effect of hold pressure time on elongation at break Source: DuPont

4.4



Hold and cooling times Source: DuPont

4.5



Schematic diagram of change in molecular structure during the hold phase Source: DuPont

4.6

Correct setting procedure

To obtain optimum moulded part properties, the hold time should be determined by the weighing method, and the cooling time should be reduced to the required minimum (which is usually just above the plasticising time). This presupposes that the gate has been correctly positioned and designed (cf. parts 2 and 3 of this series, Plastverarbeiter 46 [1995] 6 and 7). It is also important to keep the pressure constant during the hold time. The correct pressure varies between 60 and 100 MPa, depending on the material used.

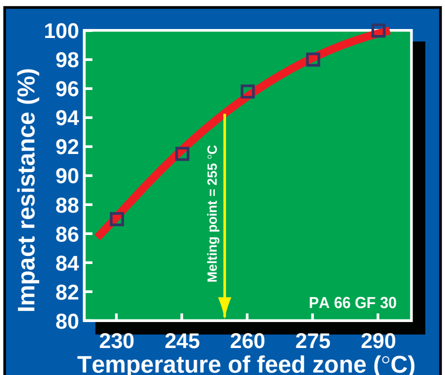
ENGINEERING POLYMERS: THE ‘TOP TEN’ MOULDING PROBLEMS

By R. Wilkinson, Roger Wilkinson, E. A. Poppe, Karl Leidig, Karl Schirmer

Chapter 5: The Wrong Melt Temperature

1. Moisture in the granules
2. Feed system too small
3. Wrong gate position
4. Hold time too short
5. Wrong melt temperature
6. Wrong tool temperature
7. Poor surface finish
8. Problems with hot runners
9. Warpage
10. Mould deposit

Choosing the right melt temperature is vital for part quality when moulding semi-crystalline engineering polymers. As a rule the margin of tolerance is less than when processing amorphous resins. The moulder at his machine directly influences the properties of the end-product. In the fifth chapter of this ten-part series, the authors consider the question of melt temperature when moulding POM (= acetal), PA (= nylon), PBT and PET (polyesters).



Influence of feed temperature
Source: DuPont
5.1

Material	Melting point	Recommended Melt Temperature °C
POM - H	175 °C	215 ± 5 °C
PA 66	255 °C	290 ± 10 °C
PA 66 GF 30	255 °C	295 ± 10 °C
PA 6	225 °C	250 ± 10 °C
PA 6 GF 30	225 °C	270 ± 10 °C
PBT	225 °C	250 ± 10 °C
PBT GF 30	225 °C	250 ± 10 °C
PET GF 30	255 °C	285 ± 5 °C

Processing temperatures
Source: DuPont
5.3

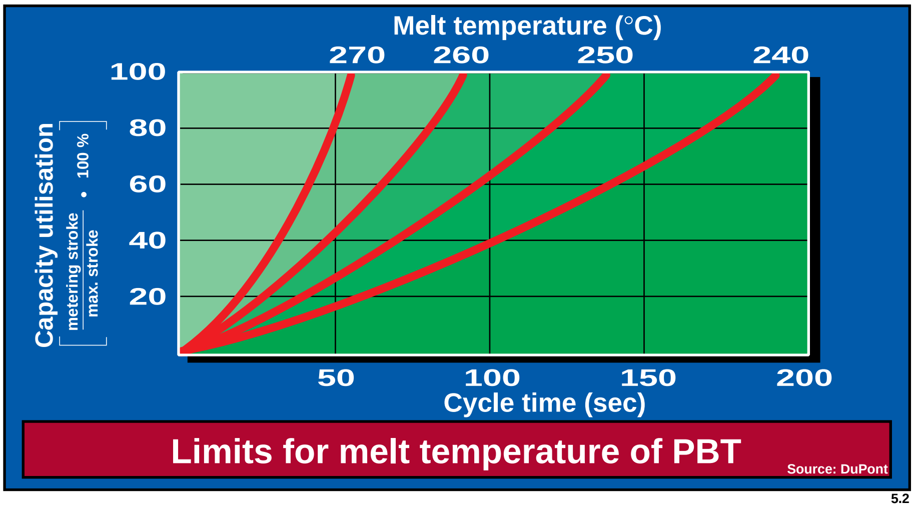
What Happens when the Melt Temperature is Wrong?

Melt temperature can be too high or too low: both are wrong. In addition, even distribution of temperature in the melt is also a factor to be kept in mind.

Temperatures that are too high degrade the polymer, that is, destroy the molecular chains. Another consequence may be that additives in the melt, such as pigments, impact modifiers, etc., also decompose. The results are poorer mechanical properties (as a result of the shorter molecular chains), surface defects (caused by decomposition products) and unpleasant odours.

When the temperature is too low, the structure fails to achieve the required homogeneity. This drastically reduces impact resistance and leads in most cases to considerable variations in physical properties.

Apart from the melt temperature, the polymer's dwell time in the injection unit also plays an important role. Experience has shown that dwell times of between two and nine minutes are normal. If the dwell time is longer, thermal decomposition may take place in certain circumstances, even if the melt temperature is correct. If the dwell time is very short, the melt usually does not have time to become fully homogeneous.



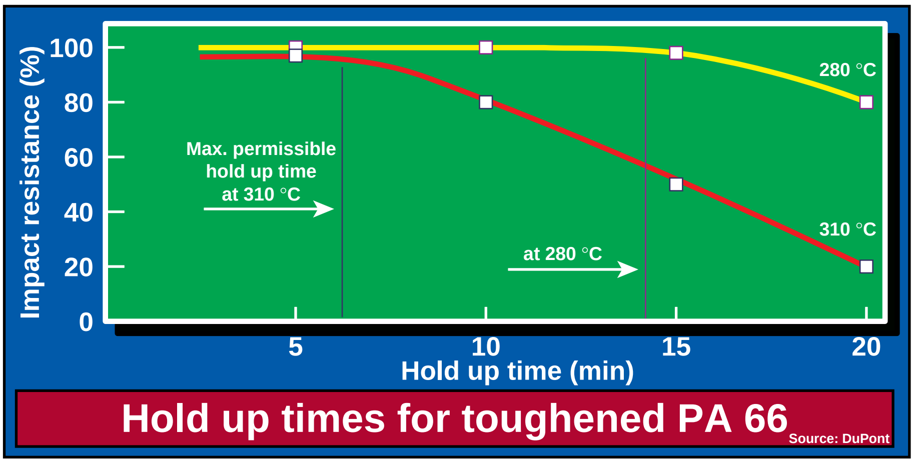
What are the Signs of Wrong Melt Temperature?

In the case of POM, excessive thermal stress generates decomposition products, causing bubbles to form in the melt. This can be observed clearly in the melt when it is purged. Other symptoms are increased mould deposit and an unpleasant odour. The physical properties of POM homopolymer are, however, hardly affected by too high melt temperatures.

PA discolours under extreme conditions, including if overheating occurs as a result of injection nozzles that are too hot. Thermal decomposition can be recognised in all PA types through reduced mechanical properties. In the laboratory, thermal decomposition can be established by measuring solution viscosity, but as a rule moulders are not in a position to apply this method.

PBT and PET react even more strongly to overheating, leading to reduced toughness. Faults are scarcely discernible during processing. If no suitable quality control measures are carried out, the damage usually becomes apparent only at the assembly stage, or when the part is in use. Discoloration indicates an unusually high degree of damage. In practice, there are tests on random samples with which certain toughness-related properties can be measured. Tests on the viscosity of moulded parts are time-consuming and expensive to carry out.

In the case of unreinforced PA or PBT, if unmelted particles are observed in the purge, it is a sign of too low melt temperature, or excessive shot size in extreme cases.



Hold up times for toughened PA 66

5.4

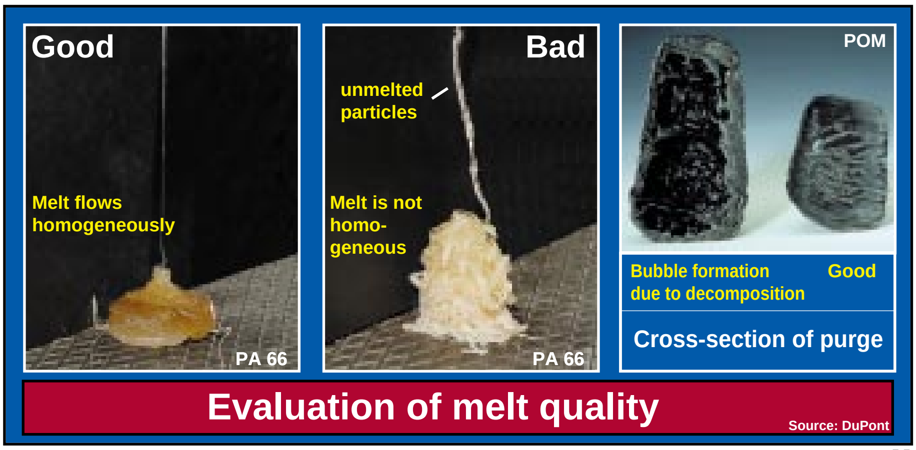
The Right Melt Temperature

The data sheets for engineering polymers indicate the optimum melt temperature range for each. In general, the temperature setting of the barrel heating zones alone is not reliable because, apart from the temperature rise due to the heater bands, friction from the screw rotation also generates heat. How much heat is generated this way depends on screw geometry and rpm as well as on back pressure.

The following recommendations can help to achieve accurate temperature measurements:

- Keep the diameter of the melt temperature probe less than 1,5 mm (response behaviour);
- Pre-heat the probe;
- Collect the melt in a thermally insulated container;
- Stir while taking measurements.

When taking initial temperature measurements or when there are no known values to rely on, a temperature profile should be selected which is 10 to 15° K above the melting point in the feed section and about 5 to 10° K under the required melt temperature in the metering zone. The temperature can be fine-tuned according to the measured melt temperature. In the case of long dwell times and short metering strokes, rising profiles are usually recommended. For short dwell times and long metering strokes flat profiles generally give the best results. A temperature zone should never be set at less than the melting point of the polymer.



Evaluation of melt quality

5.5

Measuring the temperature

5.6

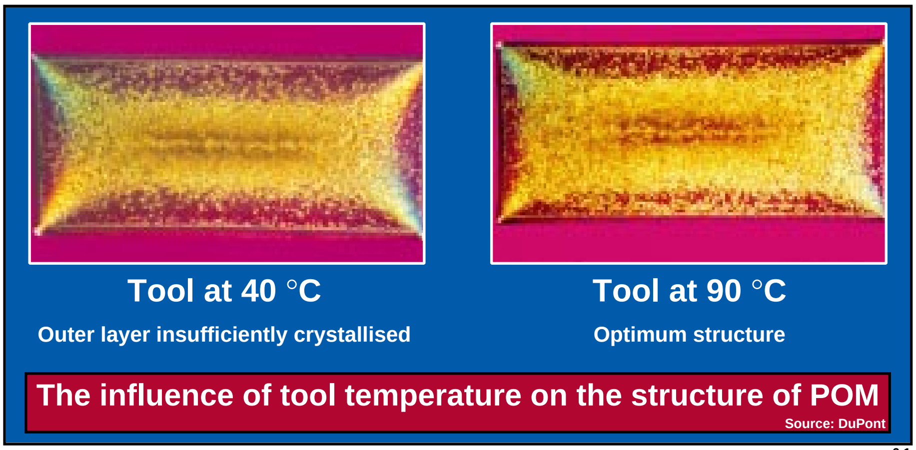
ENGINEERING POLYMERS: THE 'TOP TEN' MOULDING PROBLEMS

By R. Wilkinson, Roger Wilkinson, E. A. Poppe, Karl Leidig, Karl Schirmer

Chapter 6: Wrong Tool Temperature

1. Moisture in the granules
2. Feed system too small
3. Wrong gate position
4. Hold time too short
5. Wrong melt temperature
6. Wrong tool temperature
7. Poor surface finish
8. Problems with hot runners
9. Warpage
10. Mould deposit

When moulding semi-crystalline engineering plastics such as POM (acetal), PA (nylon), PBT and PET (polyesters), it is important to make sure that the surface temperature of the tool is correct. The basic requirements for optimum processing are in the design of the tool. Only if the tool design is right can the moulder produce good quality parts with the help of temperature control equipment. This calls for close co-operation in the tool design and planning phase, in order to avoid production problems at a later stage.



The influence of tool temperature on the structure of POM

Source: DuPont

6.1

Possible Negative Consequences of the Wrong Tool Temperature

The symptom that is easiest to recognise is poor surface finish of moulded parts. The cause is often too low surface temperature in the tool.

The mould shrinkage and post-moulding shrinkage of semi-crystalline polymers are strongly dependent on tool temperature and the wall thickness of the part. Uneven heat dissipation in the tool can thus lead to differential shrinkage. This in turn can lead to inability to maintain part tolerances. In the most unfavourable circumstances shrinkage can be beyond correction, whether working with unreinforced or reinforced resins.

When dimensions of parts in high-temperature applications become smaller with use, this is generally due to mould surface temperatures that are too low. This is because with too low mould surface temperatures mould shrinkage may be lower, but post-moulding shrinkage is substantially higher.

If a long start-up phase is needed before the dimensions settle down, it is a sign of poor temperature control in the tool, since the tool temperature is probably rising for a long time until equilibrium is reached.

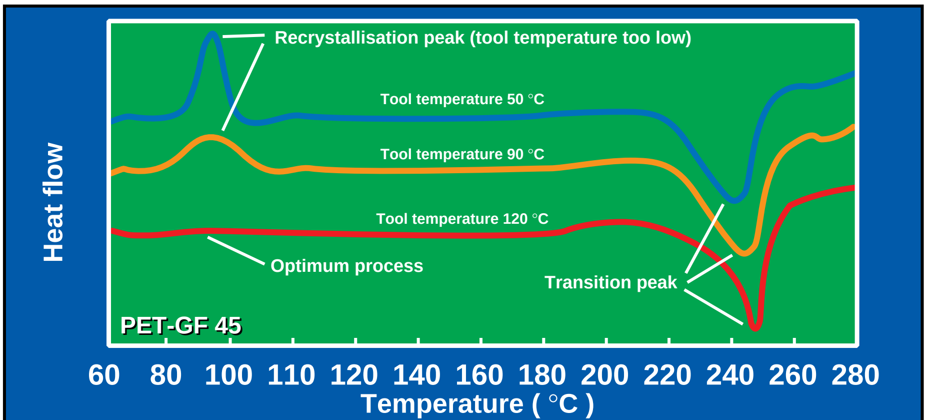
Poor heat dissipation in some regions of the tool can cause substantial lengthening of the cycle time, leading to increased cost of the moulding.

Incorrect tool temperatures can sometimes also be established from the moulded parts by means of analytical methods such as structural analysis (e.g in the case of POM) and differential scanning calorimetry (DSC) examination (e.g. with PET).

Recommendations for Setting the Correct Tool Temperature

Tools are becoming more and more complex, and as a result it is getting ever more difficult to create the proper conditions for effective mould temperature control. Except in the case of simple parts, mould temperature control systems are always a matter for compromise. For this reason, the following list of recommendations should be seen as rough guidelines only.

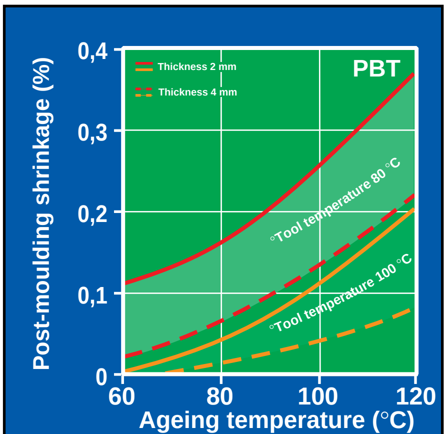
- Temperature control of the shape to be moulded must be taken into consideration at the tool design stage.
- When designing moulds that have a low shot weight and large mould dimensions, it is important to allow for good thermal transfer in the construction.
- Be generous when in dimensioning flow cross-section in the tool and in the feed pipes. Do not use fittings that cause a major restriction to the flow of the mould temperature control fluid.
- Use pressurised water as the temperature control medium, if possible. Provide flexible pipes and manifolds that are capable of withstanding high pressures and temperatures (up to 8 bar and 130° C).



The influence of tool temperature on DSC curves

Source: DuPont

6.2



Post-moulding shrinkage

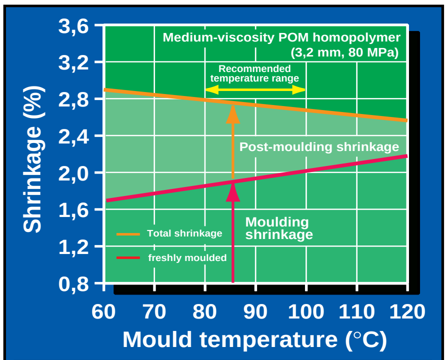
Source: DuPont

6.3

- Specify the performance of the temperature control equipment to match the tool. The tool-maker's data sheets should supply the necessary figures for flow rates.
- Use thermal insulation plates between both halves of the tool and the machine platens.
- Use separate temperature control systems for the moving half and the fixed half of the mould.
- Use separate temperature control systems for any side actions and the core, so that you can work with different start-up temperatures to get the mould running.
- Always connect different temperature control circuits in series, never in parallel. If circuits are in parallel, small differences in the flow resistance cause different volumetric flow rates of the temperature control medium, so that bigger temperature variations can occur than with circuits connected in series. (This series connection will work properly only if there is less than 5° C difference between mould inlet and mould outlet temperatures.)
- It is an advantage to have a display showing the supply temperature and return temperature on the mould temperature control equipment.
- For purposes of process control it is recommended to have a temperature sensor built into the tool, so as to be able to check its temperature during actual production.

Thermal equilibrium is established in the tool after a number of shots on cycle, normally a minimum of 10 shots. The actual temperature at equilibrium will depend on many factors. This actual temperature of the surfaces of the tool in contact with the plastic can be measured either by thermocouples within the tool (reading 2 mm from the surface) or more commonly by a hand-held pyrometer. The surface probe of the pyrometer needs to be fast-acting, and the tool temperature needs to be measured in a number of places, not just once on each side. Corrections may then be made to the set temperatures of the control units to adjust the mould temperature to what it should be. The data sheets for the various raw materials always give the recommended tool temperature. These recommendations always represent the best possible compromise between a good surface finish, mechanical properties, shrinkage behaviour and cycle times.

Moulders of precision parts and of parts that have to meet exacting optical or safety-oriented specifications generally tend to use higher tool temperatures (giving lower post-moulding shrinkage, shinier surface, more uniform properties). Technically less critical parts which have to be produced at the lowest possible cost can probably be moulded at somewhat lower tool temperatures. However, moulders should be aware of the drawbacks of this option and they should test the parts thoroughly, so as to be sure that they still meet the customer's specifications.



Shrinkage behaviour Source: DuPont

6.4

Material	Recommended tool temperature
POM - H	90 °C
PA 66	70 °C
PA 66 GF 30	110 °C
PA 6	70 °C
PA 6 GF 30	85 °C
PBT	80 °C
PBT GF 30	80 °C
PET GF 30	110 °C

Tool temperatures Source: DuPont

6.5

ENGINEERING POLYMERS: THE 'TOP TEN' INJECTION MOULDING PROBLEMS

R. Wilkinson, E.A. Poppe, K. Leidig and K. Schirmer

Part 7: Poor mould surface finish

1. Moisture in the granules
2. Feed system too small
3. Wrong gate position
4. Hold time too short
5. Wrong melt temperature
6. Wrong tool temperature
7. Poor surface finish
8. Problems with hot runners
9. Warpage
10. Mould deposit

Partially crystalline engineering thermoplastics such as POM (acetal), PA (nylon), PBT and PET (polyesters) are used primarily because of their outstanding mechanical, thermal and electrical properties. Further advantages over amorphous materials include their excellent chemical resistance and low tendency towards stress cracking. In many kinds of applications a high quality surface finish is an additional requirement. This article is intended to help eliminate possible surface defects.

Localisation and definition of surface defects

To solve the problem of surface defects one must first examine the precise location of the defect, and when it actually became evident. Here it is advisable to observe the surface during the actual injection moulding process. The points that need clarification are listed below.

- Does the defect occur with every shot or irregularly?
- Does the defect always occur in the same cavity?
- Does the defect always occur at the same place in the moulding?
- Can the defect be predicted already during a mould filling study?
- Is the defect already evident on the sprue?
- How does the defect react when a new batch of moulding compound is used?
- Does the defect occur with only one machine or with others as well?

Analysis of possible causes of surface defects

Surface defects may be caused by many different factors such as:

- Compounding: drying, compound quality, presence of contaminants (foreign bodies)
- Injection moulding conditions: melt temperature, injection speed and change-over point
- Condition of the injection unit, e.g. wear and dead spots
- Design of the hot runner system (runners, material stoppage etc.)
- Mould design, position of gate and gate cross-section, cold slug interceptor, venting etc.
- Additives such as pigments
- The polymer contained in the moulding compound.

Conclusions to be drawn from surface defects

1. Regular local defects

If surface defects regularly occur in the same place this indicates that there is a problem in the injection nozzle or the hot runner nozzle. The shape and design of the runner, gate or the moulded part itself may be responsible, e.g. sharp edges, sudden changes in wall thickness etc. Another cause may be moulding conditions such as the injection profile or the change-over point.

2. Irregular local defects

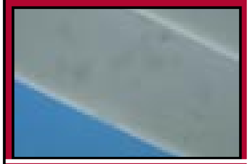
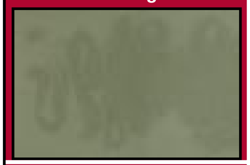
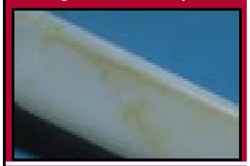
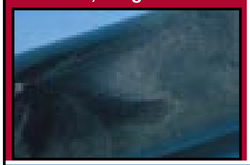
Where surface defects occur irregularly in different places, one should look at compounding (compound quality, presence of dust). Factors such as low melt temperature, back pressure, screw speed and screw retraction can also play an important part.

3. Surface defects covering large areas

This kind of defect usually extends over the entire moulding and is often visible already on the sprue. Here one should check whether melt decomposition has occurred. This is done by forcing a shot of melt into the open and observing whether, for example, it contains bubbles. In the case of hot runner systems, this method can be used with only limited success. Melt decomposition may be due to polymer degradation or decomposition of additives, caused by overheating or excessively long residence times. In the case of hygroscopic polymers, an important part is also played by hydrolytic degradation if the moulding compound has not been dried sufficiently.

General recommendations

Parts made from partially crystalline engineering polymers should preferably not be made by hot runner injection moulding if a perfect surface finish is essential. It is advisable to make use of a subsidiary runner, which thermally isolates the nozzle from the moulded part, thereby reducing the risk of surface defects.
The cold slug coming from the injection or hot runner nozzle should be intercepted by a special device opposite the sprue so that it cannot get into the moulded part.
The following table lists various surface defects and ways and means of eliminating them. In practice however, different surface defects appear simultaneously, which makes any investigation as to their origin – and their elimination – much more difficult.

Symptom	Grades	Where and when	Possible cause	Possible elimination
Streaks in direction of flow 	all	• with every shot across large areas	• damp compound (PA) • thermal degradation	• check moisture content of compound • check drying • check melt temperature
Marbling 	mineral reinforced grades	• with every shot • behind sharp edges • near the gate	• too high shear • slip-stick effect (outer skin migration)	• reduce injection speed (possibly profile) • round off sharp edges • increase gate cross-section
Cold slug 	all, especially reinforced grades	• usually occurs in only one place • goes through the entire wall thickness	• cold slug or inhomogeneous melt from the injection or hot runner nozzle has got into the moulded part	• intercept cold slug • possibly raise nozzle temperature
Sink mark 	all, especially unreinforced grades	• opposite ribs • near melt accumulations	• greater shrinkage near melt accumulations since the holding pressure here is not effective long enough	• improve design, e.g. make ribs thinner and provide uniform wall thickness • move gate elsewhere
Charred surface 	all	• always at the same place (near weld lines and at the end of flow paths)	• oxidation through compressed air which cannot escape (Diesel effect)	• provide or improve venting • inject more slowly
Unmelted particles 	all grades, especially unreinforced ones	• sporadically in different places	• compound has not melted and is not properly homogenised	• check melt temperature (perhaps too low) • increase back pressure • check screw speed • possibly use bigger cylinder (longer residence time)
Jetting 	all	• with every shot • usually starting from the gate	• jet of melt issues from the gate into the moulded part • no flow resistance to support laminar flow	• inject more slowly (possibly profile) to obtain laminar flow • provide flow restrictor behind the gate • possibly move gate elsewhere
Irregular brown spots 	all	• 5-15 shots are correct, then defect occurs during the next 1-2 shots, to be followed by another 5-15 good shots etc.	• dead spot in nozzle or hot runner (e.g. ante-chamber). Only when the compound has degraded is it forced into the melt stream. Dead spot is then filled with fresh melt etc.	• remove dead spot • improve bypass
Whitish, rough surface 	reinforced grades	• near the end of the flow path • behind edges and bypasses • near ribs	• melt front stops for short time during filling • polymer crystallises before it is forced against the wall • glass fibres near the surface	• increase injection speed • check melt temperature, which may be too low • check change-over point and mode (do not fill with holding pressure)

Typical surface defects and their elimination

Source: DuPont

7.1

ENGINEERING POLYMERS: THE ‘TOP TEN’ INJECTION MOULDING PROBLEMS

R. Wilkinson, E.A. Poppe, K. Leidig and K. Schirmer

Part 8: Problems with hot runners

1. Moisture in the granules
2. Feed system too small
3. Wrong gate position
4. Hold time too short
5. Wrong melt temperature
6. Wrong tool temperature
7. Poor surface finish
8. Problems with hot runners
9. Warpage
10. Mould deposit

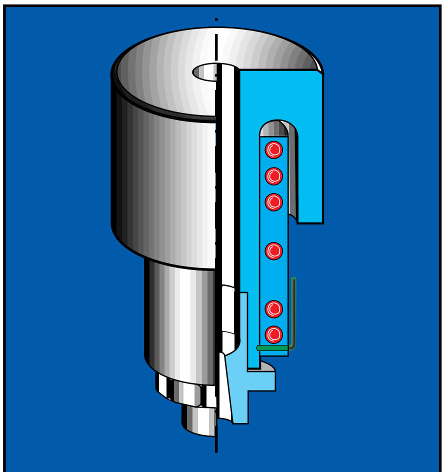
When injection moulding partially crystalline engineering thermoplastics, choice of the correct hot runner system determines the function of the mould and moulded part quality. Here, the temperature must be controlled much more strictly than in the case of amorphous materials. The type of hot runner system used, and its installation, decide the properties of the finished parts. This article deals with the most important points which have to be considered when choosing the most suitable hot runner system for POM (acetal), PA (nylon), PBT and PET (polyesters).

What happens when an unsuitable hot runner system is used?

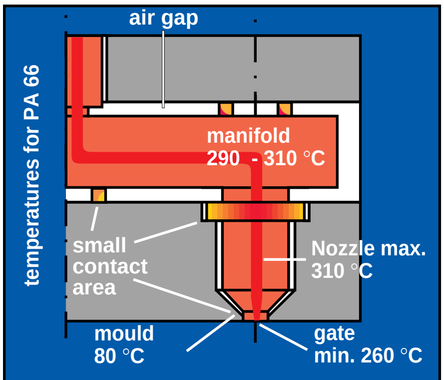
Unsuitable hot runner systems usually cause high pressure losses; they should, if used at all, be operated only at very high temperatures. This will usually cause the polymer to degrade, with all the consequences already described in part 5 of this series of articles, entitled “Wrong melt temperature”. Streaks, discoloration and surface defects will also be produced, due to local overheating. The resultant decomposition of the moulding compound causes blistering and other undesirable effects due to degradation products.

What points should be considered?

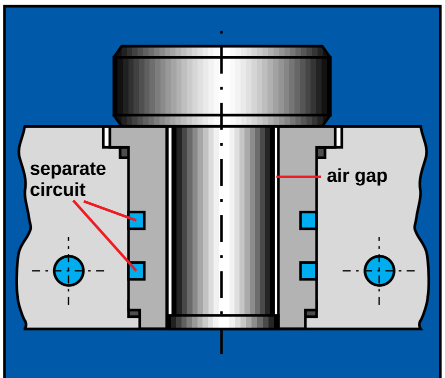
The above-mentioned polymers all display a certain amount of latitude between recommended melt temperature and solidification temperature. It is therefore necessary to effectively thermally isolate the hot runner from the runners and nozzles.
The nozzles should be designed so that naturally balanced runners can be used. This is the only way of ensuring uniform pressure losses and the same melt residence time in all the mould cavities.
In the case of small shot weights, indirect gating is preferable to direct gating, especially with glass fibre reinforced materials. Material throughput per nozzle increases, so that the heat applied to the moulding compound is more easily manageable. Gates for hot runner nozzles can be large, the gate on the moulded part remaining small through conventional gating. A cold slug interceptor should in any case be arranged facing the hot runner nozzle. This is the only way of preventing cold material getting into the moulded part through the nozzle.



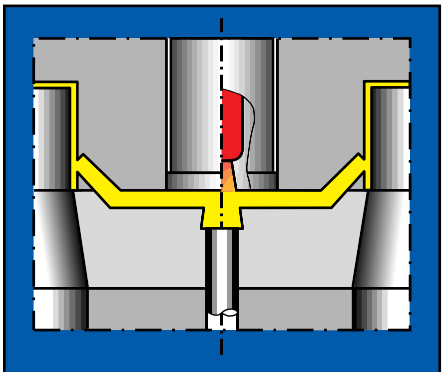
Recommended nozzle design
Source: DuPont
8.1



Thermal isolation
Source: DuPont
8.2



Separate heating
Source: DuPont
8.3



Indirect gating
Source: DuPont
8.4

Separate controls should be provided for the hot runner inlet, the runner and each nozzle, to enable all parts to be individually balanced with regard to the thermally sensitive moulding compounds. Regulating devices should be used which guarantee constant temperatures through adaptation of the power supply (e.g. PID).
The hot runner system should be mechanically supported in the same way as the ejector system. The mould is weakened near the runners and this must be compensated as much as possible. Separate heating circuits in the immediate vicinity of the hot runner nozzles allow mould surface temperatures to be correctly set independently.

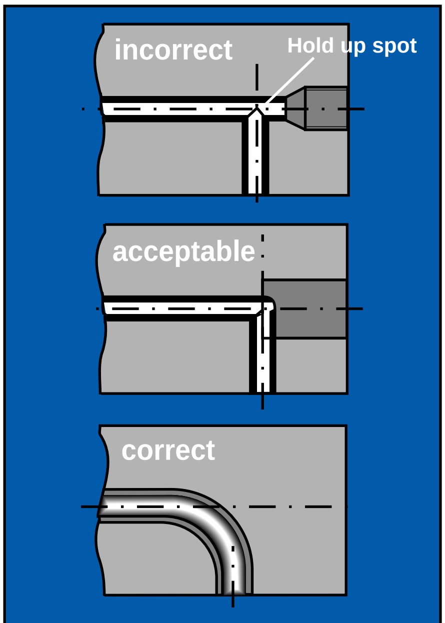
Selection criteria for hot runners and nozzles

Runners with a full cross-section and symmetrically incorporated heat conductors are the best solution. Internally heated systems which only have an annular cross-section cause excessively high pressure losses and should be avoided if at all possible.

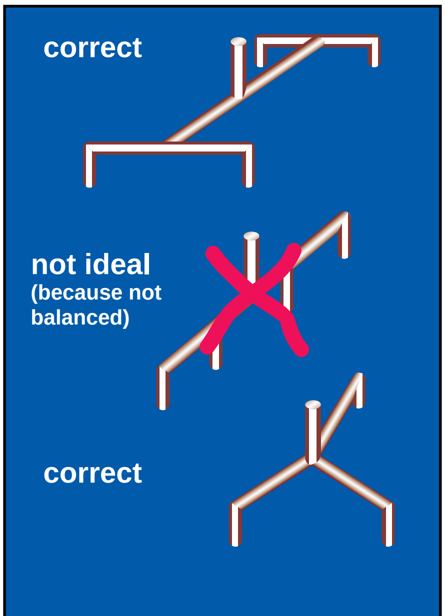
In the case of materials with high thermal sensitivity such as POM and flame resistant compounds, the bypass inside the runner should be as perfect as possible.

Nozzles should be open, externally heated systems with a full cross-section. Division of the melt stream into several streams should be avoided in the gate region. Distribution of the connected load should be adapted to conditions in the built-in state, so that there is even temperature distribution. It is an advantage to provide interchangeable nozzle tips in case of abrasive materials having to be processed. In addition, compromises with small torpedoes are possible if necessary.

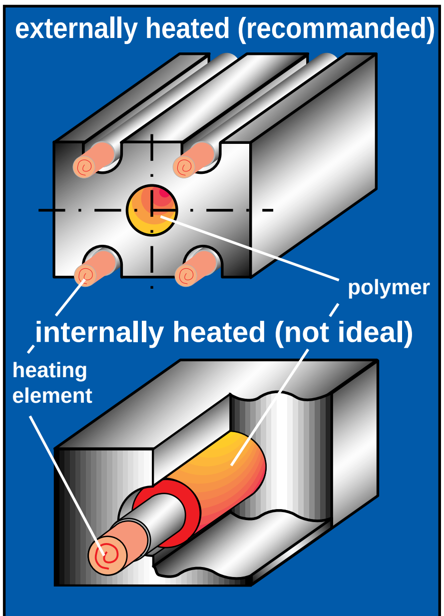
It is generally inadvisable to use shut-off nozzles when processing POM. If the use of other kinds of compound dictates the use of needle valve nozzles, nozzle/needle combinations should be used which keep pressure losses as low as possible. Many different kinds of hot runner systems are on the market, which give excellent results provided the above recommendations are followed.



Turn of melt stream
Source: DuPont
8.5



Nozzle arrangement
Source: DuPont
8.6



Flow cross-section inside manifold
Source: DuPont
8.7

ENGINEERING POLYMERS: THE 'TOP TEN' INJECTION MOULDING PROBLEMS

R. Wilkinson, E.A. Poppe, K. Leidig and K. Schirmer

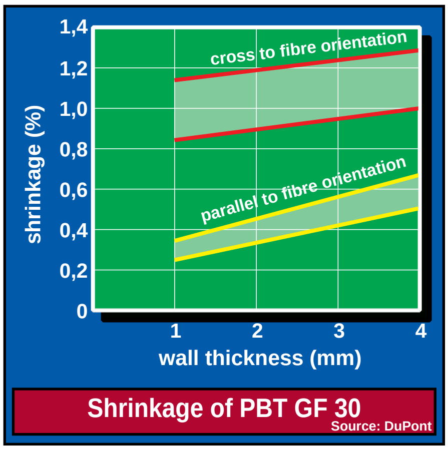
Part 9: Warpage

1. Moisture in the granules
2. Feed system too small
3. Wrong gate position
4. Hold time too short
5. Wrong melt temperature
6. Wrong tool temperature
7. Poor surface finish
8. Problems with hot runners
9. Warpage
10. Mould deposit

Partially crystalline substances such as POM (acetal), PA (nylon), PBT and PET (polyesters) tend to warp far more than amorphous ones. This point should be taken into consideration already when designing moulds and mouldings. If this is not done, it is almost impossible to rectify at a later stage. This article discusses the causes of warpage and steps that can be taken to prevent and reduce it.

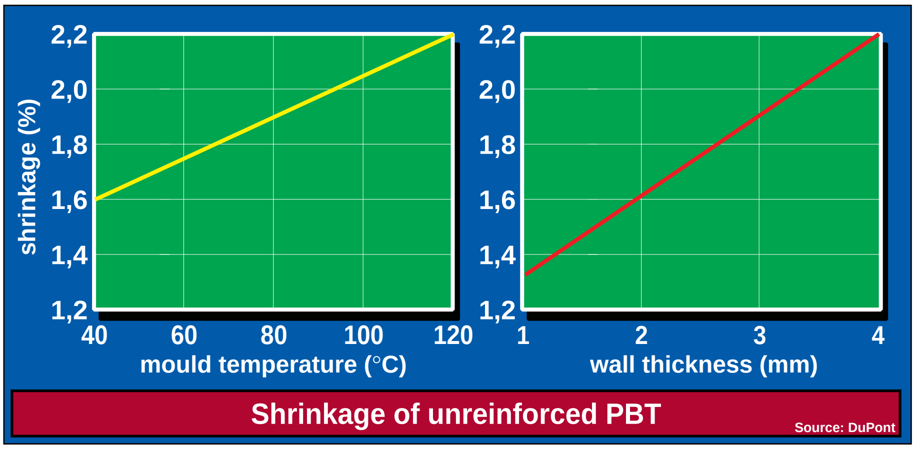
What are the main causes of warpage?

Shrinkage is relatively high in partially crystalline materials and is influenced by a number of factors. In the case of unreinforced materials, warpage is greatly influenced by wall thickness and mould surface temperature. It follows that major differences in wall thickness and unsuitable mould temperatures will cause the moulding to warp. Totally different shrinkage characteristics will be evident in the case of glass fibre reinforced materials, due to orientation of the glass fibres. The effect of wall thickness differences on shrinkage is relatively slight. Here, the main cause of warping is the difference between fibre orientation longitudinally and at right angles to the direction of flow. Warpage is essentially due to wall thickness distribution, gate location, flow restrictions and by-passes, as well as the inherent rigidity of the moulded part.
These different causes of warping, depending on whether the material is fibre-reinforced or not, frequently result in contrary warping phenomena in the same part.



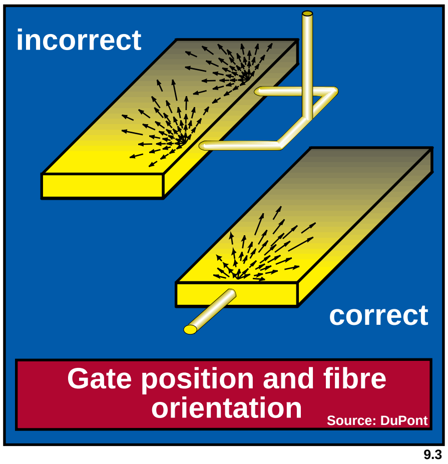
Shrinkage of PBT GF 30

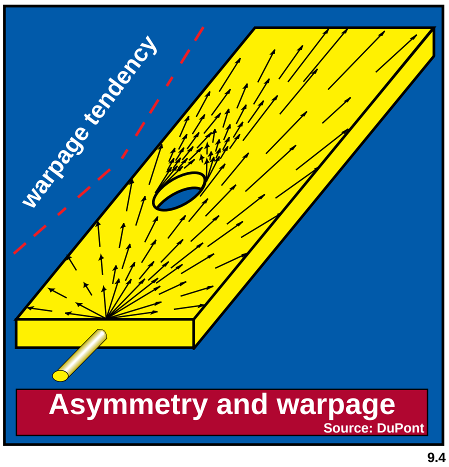
9.1

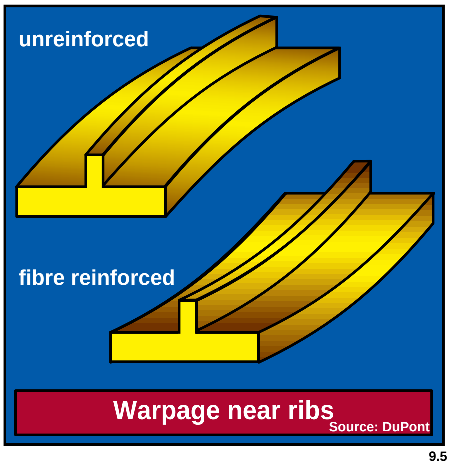


Shrinkage of unreinforced PBT

9.2





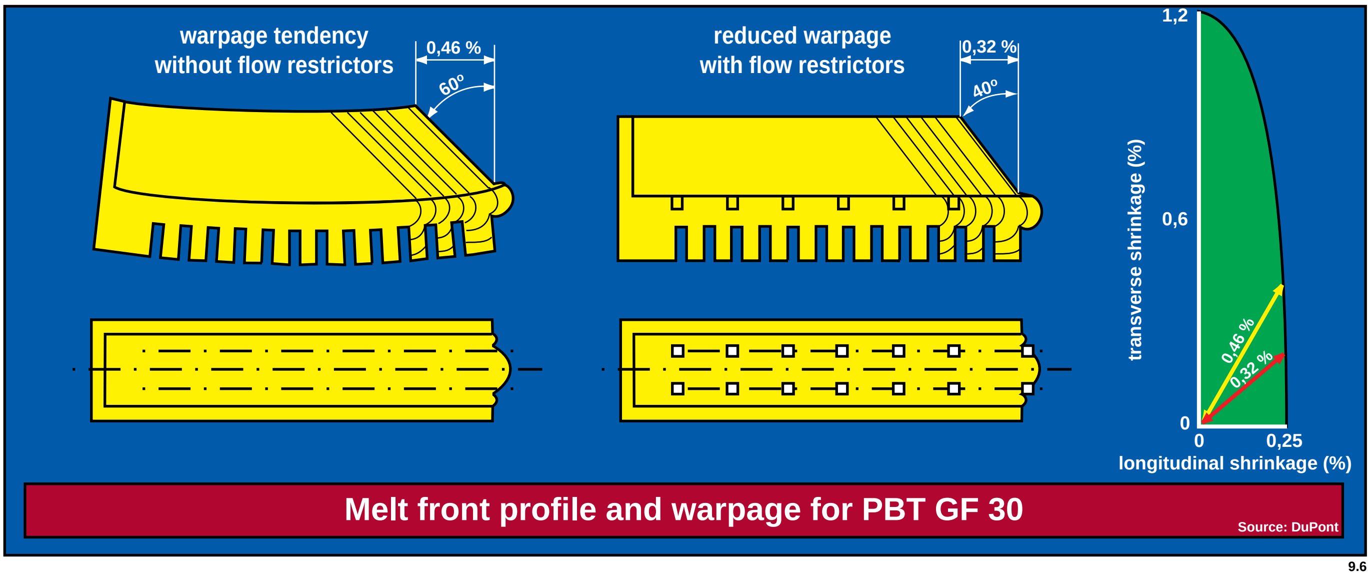


How can warpage be prevented?

Unreinforced materials require uniform wall thicknesses. Melt accumulations should be avoided as far as possible. Multi-point gating can be used to achieve a high pressure gradient and thus reduce shrinkage differences to a minimum. The mould heating system should be designed so that heat is dissipated as evenly as possible (see No. 6 of this series of articles).
With glass fibre reinforced materials, the symmetry of the moulded part is as important as uniform wall thickness. Asymmetrical parts hinder melt flow and thus orientation, and eventually cause warpage. In the case of asymmetrical parts it is therefore necessary to achieve a balance by incorporating blind cores at the mould planning and design stage. The position of the gate is also important – every by-pass and every weld line is a potential cause of warping.

What possibilities are open to the moulder?

Assuming that the moulded part, the gate and the mould have all been correctly designed, the moulder can control warpage up to a point via the holding pressure and mould temperature. The use of several heating circuits to optimise heat dissipation is normal practice.
In the case of reinforced materials, changing the injection rate or lowering the mould temperature is a slight help. If the possibility of subsequent warpage has not been foreseen at the mould and moulded part design stage, this cannot be subsequently rectified by modifying moulding conditions.



What can be done when warping has occurred?

The most important step, especially in the case of glass-fibre-reinforced materials, is to carry out a mould filling study, i.e. by partly filling the mould in several stages. By studying the melt front profile, fibre orientation can be reconstructed. By referring to the shrinkage curve for the reinforced material, measures can be taken to reduce warpage, e.g. by incorporating flow aids or flow restrictors. These alter the melt front profile, thereby influencing warpage.

This method requires a great deal of practical experience and, at the same time, increases the amount of knowledge to enable precautionary measures to be taken in future. It also has its limitations, due to the properties of the raw material and physical conditions. For crystalline polymers, it is not possible to obtain the same flatness as for amorphous polymers. In this connection, mention should be made of low-warpage, semi-crystalline polymer blends. These represent a compromise between properties and warpage, due to chemical modification or the combination of different reinforcing components. The last, and also the most expensive method consists of modifying the mould. If there already is experience with similar mouldings, correctable inserts are the best solution for critical parts.

ENGINEERING POLYMERS: THE 'TOP TEN' INJECTION MOULDING PROBLEMS

R. Wilkinson, E.A. Poppe, K. Leidig and K. Schirmer

Part 10. Deposits on mould surface.

1. Moisture in the granules
2. Feed system too small
3. Wrong gate position
4. Hold time too short
5. Wrong melt temperature
6. Wrong tool temperature
7. Poor surface finish
8. Problems with hot runners
9. Warpage
10. Mould deposit

These can occur with nearly all thermoplastics. As demands on end-products increase, so do the amounts of additives which have to be incorporated, e.g. modifiers, flame retardants, etc. These additives may quite often cause deposits to form on the mould cavity surface.
There are many other associated reasons for the formation of mould deposits. The most common ones are
• thermal decomposition
• excessive shear
• inadequate venting.
Such deposits are often due to a combination of different factors and a great deal of trouble has to be taken to find out exactly what causes these deposits to form and how to prevent them. One problem is that deposits often build up only after a few days.

Types of deposit

Each group of additives produces a specific type of deposit. Flame retardants can react at high temperatures, forming decomposition products which may produce deposits. Impact modifiers are affected not only by excessively high temperatures but also by excessive shear. Modifiers can, under unfavourable conditions, separate from the polymer and form deposits on the cavity surface.
Pigments in engineering thermoplastics needing high melt temperatures, can reduce the thermal stability of the moulding compound, resulting in deposits consisting of polymer degradation products and decomposed pigments.

In parts of the mould which become especially hot (such as cores), modifiers, stabilisers and other additives may stick to the surface and build up deposits. In such cases, steps must be taken to achieve better mould temperature control or use special stabilisers. The table lists the possible causes of mould deposits and ways and means of preventing them.

	Possible causes	Possible elimination
Thermal decomposition	Melt temperature too high, residence time too high	• measure melt temperature and reduce to recommended level • check ejected compound for signs of decomposition, e.g. formation of bubbles, or gas in the melt • match cylinder temperature to residence time • ensure thermal insulation of hot runner, check temperature control and lower temperature
Thermal decomposition	Dead spots in nozzle, near non-return valve, wear in cylinder, dead spots in hot runner	• show up dead spots through changing colour. Long cleaning cycles mean poor purging • examine suspicious components (nozzle, adapter, screw, hot runner) for dead spots and repair or exchange
Thermal decomposition	Polymers or additives with insufficient thermal stability	• reduce residence time by using smaller cylinder. Operate with delayed feed. Ensure small melt cushion. Keep melt decompression to a minimum to prevent oxidation through sucked-in air. • use standard product (without modifiers, pigments etc.) experimentally. • pre-dry to reduce volatile constituents
Too high shear	Walls too thin or flow distance too long, resulting in high shear stress	• make walls thicker or incorporate flow aids • increase number of gates to reduce flow distances • change gating system, possibly use hot runner. • possibly increase melt temperature
Too high shear	High shear due to too small a gate	• increase gate cross-section • redesign gate • increase number of gates
Too high shear	High shear due to fast injection	• reduce or profile injection fill speed • increase melt temperature
Insufficient venting		• provide cavities with vents or improve venting system • fit self-cleaning vents to ensure consistent removal of air.
Mould surface temperature too high		• measure mould temperature after the starting-up phase and reduce to recommended level • reduce overheating of cores by adjusting mould temperature control

Possible causes and remedies for mould deposits

Source: DuPont

10.1

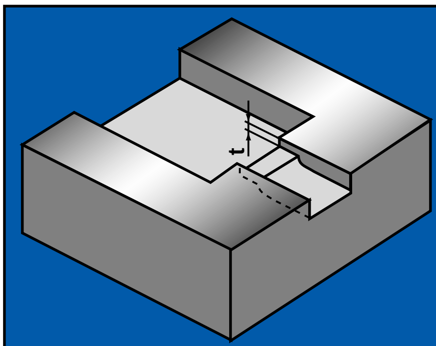
Deposits which appear suddenly

If deposits appear suddenly, this may be due to changed moulding conditions, or the moulding compound when batches are changed. The following comments may be helpful.

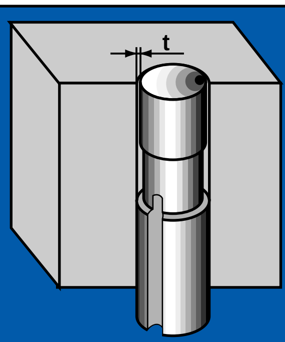
First of all, the melt temperature should be measured and the melt visually checked for signs of decomposition, i.e. the presence of charred particles. One should also check whether the moulding compound has become contaminated by foreign substances and that no incompatible purging compound has been used. Mould venting should also be checked. The next step should be to run the machine with natural or pastel-coloured moulding compound (but not black). The machine is then switched off after about 20 minutes' moulding. Nozzle, adapter and possibly the screw are then dismantled. Inspection of the material for charred particles, and comparing its colour with that of the original moulding compound will quickly indicate the source of the problem.
This technique has in many instances uncovered surprising weaknesses, but is really applicable only to small machines (up to, say, 40 mm screw diameter). Their elimination will also result in definite quality improvements when processing other materials. A similar procedure can be adopted for hot runner systems.

Care of moulds

If deposits cannot be prevented from forming by any of the measures described above, special care and attention will have to be given to the moulds.
It has been found that deposits on the mould surface can be removed relatively easily in the early stages. Cavities and vents should therefore be cleaned at specified intervals, e.g. at the end of each shift. Once the deposit has formed a thick layer it is extremely difficult and time-consuming to remove it.
Because deposits vary so widely in their chemical composition, trials have to be carried out to find the most suitable solvent which will shift them. Besides the classic solvents, it has often been found that unconventional products can solve the problem, e.g. oven sprays or lemonades containing caffeine. Another trick is to use cleansing rubbers as used in model railways.



in mould parting surface



using ejector

Material	t*
POM	0,03 mm
PA	0,02 mm
PET	0,02 mm
PBT	0,02 mm
TEEE	0,03 mm

* in the case of low viscosity grades and where there must be a minimum amount of flash, vent slits should be shallower to start with

Recommendation for vent depth

Typical vent designs

Source: DuPont

10.2

Recommendations on preventing deposits

If thermally sensitive compounds are moulded using hot runners, it should be remembered that the residence time will be longer, so that the risk of deposits consisting of degradation products will be greater.
Shear sensitive materials should always be processed using generously dimensioned runners and gates. Multi-point gating, which reduces flow distances and thus enables moulders to reduce injection speeds, have given good results.
Generally speaking, efficient mould venting reduces the tendency to form deposits. Adequate venting should therefore be provided at the mould design stage. Self-cleaning vents, or those from which deposits can be easily removed, are to be preferred. Improvements in the venting system have often led to reduced deposits on the tool.
In some instances it is possible to apply a special non-stick coating to the cavity surface, which will prevent deposits building up. Tests should be carried out to assess the effectiveness of such coatings. Titanium nitriding has often reduced the rate at which deposits build up on the tool.

Deposit on overheated core

Source: DuPont

10.3