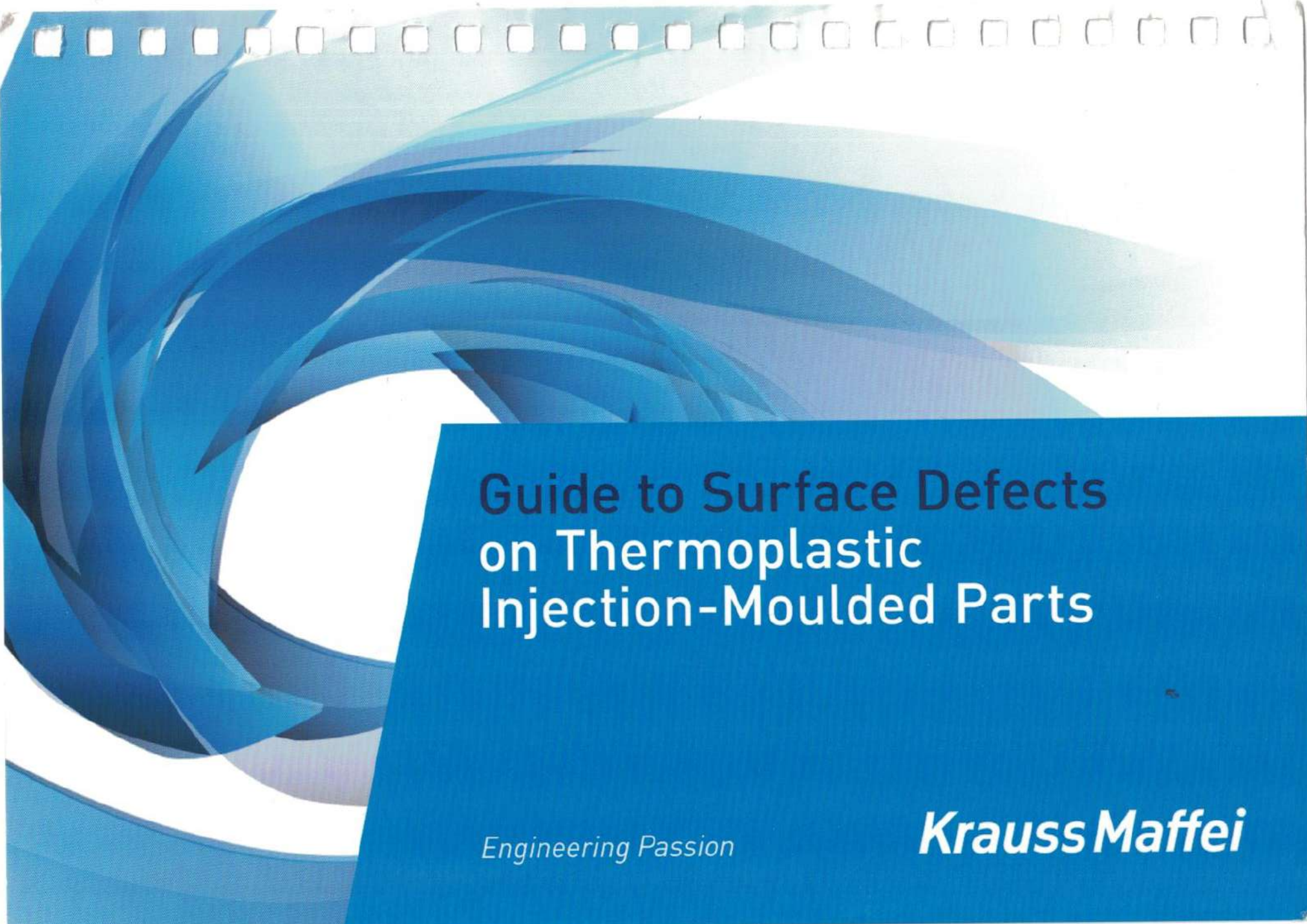




Guide to Surface Defects on Thermoplastic Injection-Moulded Parts

Engineering Passion

Krauss Maffei



GUJARAT STATE
PLASTICS
MANUFACTURERS
ASSOCIATION

**With a Kind courtesy for the
Injection Moulding Industries
OFFERED BY**



Mr. Dirk Vermeer,
Volunteer Professional of PUM Netherlands

**11th Edition, Lüdenscheid
May 2013
Copyright K.I.M.W. NRW GmbH**

This work is protected by copyright. All rights reserved, including translation, reprint and duplication of the book or parts thereof.

No part of this work may be electronically processed, copied or distributed in any form (photocopy, microfilm or any other procedure), not even for the purpose of teaching, without written consent by the Kunststoff-Institut Lüdenscheid.

Kunststoff-Institut Lüdenscheid
K.I.M.W. NRW GmbH
Karolinenstr. 8
D-58507 Lüdenscheid

Phone: +49 (0) 23 51.10 64-191
Fax: +49 (0) 23 51.10 64-190
E-mail: mail@kunststoff-institut.de
Internet: www.kunststoff-institut.de

Scientific involvement:
Kunststoff-Institut für die mittelständische
Wirtschaft NRW GmbH with the
support of numerous partners from the
industry

Realization:
Horschler Kommunikation GmbH,
Unna, Germany

Print: DruckVerlag Kettler, Bönen,
Germany

Publisher:
Horschler Verlagsgesellschaft mbH
Friedrich-Ebert-Straße 19
59425 Unna, Germany
Phone: +49 (0) 23 03.2 54 22-0

ISBN: 978-3-9810961-4-9

The new "Guide to Surface Defects on Thermoplastic Injection-Moulded Parts" is an extensive revision and extension of our first "Guide to Surface Defects", of which more than 100,000 copies in 11 editions have been distributed worldwide in six different languages since 1992.

This edition has been largely extended in view of the latest technologies. The following defects have been added to the ones previously described:

Defects of thermoplastic foam injection moulded parts (TSG parts), unevenly filled parts, and possibilities of dynamic temperature control of the mould in order to

avoid surface defects. In addition, the guide was extended by the chapters concerning the effect of parameter changes on the quality of the moulded parts and general notes on the identification and correction of defects.

An additional index and a fold-out table of contents enhance the orientation in the guide.

**Kunststoff-Institut
Lüdenscheid
April 2013**

Chapter 1:

Defect classification (identification)

This chapter describes the different defects with the help of numerous pictures and explanations. In this way, defects can be identified and classified.

Chapter 2:

Physical causes and remedies

The possible physical causes of each defect are described by way of brief explanations. Important notes regarding the influencing factors to be considered are also supplied.

Flow diagrams then provide information on eliminating or reducing defects. Apart from process-related remedies there are also notes on the design of the moulded part and the mould as well as on the material.

Chapter 3:

Effect of parameter changes on the process and on the quality of the moulded part

For the most important process-related, adjustable parameters, the influence or effect on other process parameters and – as

a tendency – also on certain quality features is represented in tabular form.

Chapter 4:

General notes on the identification and correction of defects

The detailed information concerning the identification and elimination of defects in the first two chapters is now followed by additional, helpful tips concerning troubleshooting and the fundamental process settings.

Chapter 5:

General processing instructions

This chapter provides guide values for certain processing parameters, e.g. processing temperature, screw speed, back pressure, and an overview concerning material combinations for the multi-component process.



Engineering Passion

KraussMaffei

Introduction ↓

Introduction

The structure of this guide is based on a method that you should apply for the optimisation of your processes.

The initial visual inspection of the moulded part is followed by the identification and classification of defects in order to initiate suitable counter-measures. For this purpose, however, the fundamental procedures for systematic mould proving must be known and clearly defined. Another aid is provided in the form of a list of important key information concerning the technology of materials, machines, moulds, and peripheries. This guide is, therefore, divided into five chapters:

Chapter 1:

Defect classification (identification)

Page 6 – 49

Chapter 2:

Physical causes and remedies

Page 50 – 148

Chapter 3:

Effect of parameter changes on the process and on the quality of the moulded part

Page 150 – 157

Chapter 4:

General notes on the identification and correction of defects

Page 158 – 169

Chapter 5:

General processing instructions

Page 170

table of contents ↓

Table of contents Chapter 1 and 2 - numerical

No.	Topic	Page	
		Cause	Remedy
1.	Sink marks	7	53
2.	Streaks	8	55
2.1	Charred streaks	11	55
2.2	Moisture streaks	12	57
2.3	Air streaks	13	59
2.4	Coloured streaks	14	61
2.5	Glass fibre streaks	16	63
3.	Blistering	17	65
3.1	Air entrapments	18	66
3.2	Voids	19	67
3.3	Gaseous entrapments	20	69
4.	Weld line	21	71
5.	Gloss/gloss differences	22	75
6.	Jetting	23	80
7.	Record grooves effect	24	82
8.	Dull spots near the gate	25	84
9.	Incompletely filled parts	26	86
10.	Diesel effect/burn	27	88
11.	Overpacked parts (flashes)	28	90
12.	Stress-whitening/stress cracks	29	92
13.	Visible ejector marks	30	94
14.	Deformation during demoulding	31	96
15.	Ejection grooves	32	100
16.	Flaking of the surface layer	33	102
17.	Cold slugs	34	104
18.	Filamentation	35	106
19.	Dark spots	36	108
20.	Plate-out	37	111
21.	Broken/ragged film hinge	38	114
22.	Warpage	39	116
23.	Multi-component technology	40	124
24.	Tiger lines	41	129
25.	Defects of thermoplastic foam injection moulded parts	42	132
26.	Unevenly filled parts/flow anomalies	47	140
27.	Dimensional and weight variations	49	145

Table of contents Chapter 1 and 2 - alphabetical

Topic	No.	Page	
		Cause	Remedy
Air entrapments	3.1	18	66
Air streaks	2.3	13	59
Blistering	3.	17	65
Broken/ragged film hinge	21.	38	114
Charred streaks	2.1	11	55
Cold slugs	17.	34	104
Coloured streaks	2.4	14	61
Dark spots	19.	36	108
Defects of thermoplastic foam injection moulded parts	25.	42	132
Deformation during demoulding	14.	31	96
Diesel effect/burn	10.	27	88
Dimensional and weight variations	27.	49	145
Dull spots near the gate	8.	25	84
Ejection grooves	15.	32	100
Flaking of the surface layer	16.	33	102
Filamentation	18.	35	106
Gaseous entrapments	3.3	20	69
Glass fibre streaks	2.5	16	63
Gloss/gloss differences	5.	22	75
Incompletely filled parts	9.	26	86
Jetting	6.	23	80
Moisture streaks	2.2	12	57
Multi-component technology	23.	40	124
Overpacked parts (flashes)	11.	28	90
Plate-out	20.	37	111
Record grooves effect	7.	24	82
Sink marks	1.	7	53
Streaks	2.	8	55
Stress-whitening/stress cracks	12.	29	92
Tiger lines	24.	41	129
Unevenly filled parts/flow anomalies	26.	47	140
Visible ejector marks	13.	30	94
Voids	3.2	19	67
Warpage	22.	39	116
Weld line	4.	21	71



Chapter 1

Defect classification (identification)

CHAPTER 1

Defect classification
(identification)



1.1 Sink marks

Sink marks appear, for example, near material accumulations in the form of depressions on the surface of the moulded part, if the thermal contraction (shrinkage) cannot be compensated for.



1.1

Sink marks near ribs



1.2

Sink marks near a dome



1.3

Sink marks on a cylindrical core with poor temperature control



Help on classifying streaks:

Streaks, especially charred streaks, moisture streaks and air streaks tend to look very similar. This is a problem for production in particular, since visual identification is difficult, if not impossible. The processor therefore needs as much information as possible on the plastic, its processing and also on environmental influences, in order to identify the streak. This chapter will therefore provide some information on the different types of streak to facilitate their classification.

Not all of the characteristics mentioned will necessarily have to appear.

CHAPTER 1

Defect classification
(identification)



Help on page 55



1.2 Characteristics of streaks

Charred streaks

- ▶ The streak appears periodically.
- ▶ The streak appears behind narrow cross-sections (shear points, e.g. gating system) or behind sharp edges in the mould.
- ▶ The melt temperature of the injected melt is near the upper processing limit.
- ▶ Lowering the injection speed reduces the defect.
- ▶ Lowering the melt temperature reduces the defect.
- ▶ The melt resides in the plasticising unit or the space in front of the screw for a long period of time, e.g. due to cycle breaks or low shot volumes.
- ▶ The reclaim content is high, or a part of the material has already been melted several times before.
- ▶ The mould is equipped with a hot-runner.
- ▶ The mould is equipped with a shut-off nozzle.
- ▶ The moulding compound resides in the dryer for too long.

Moisture streaks

- ▶ The material tends to absorb moisture (e.g. PA, ABS, CA, PBT(P), PC, PMMA, SAN).
- ▶ During slow injection "to the outside", the melt is steaming and/or showing blisters as it comes out of the nozzle.
- ▶ The solidified flow front of a partial filling shows crater-like structures.
- ▶ The moisture content of the material prior to processing is very high.
- ▶ The moisture content in the environment is very high.
- ▶ The mould wall is very cold.
- ▶ Especially when combined, the last two issues lead to moisture streaks.
- ▶ The temperature at the material feed (flange temperature) is very low.
- ▶ The mould cooling system is leaking.



Air streaks

- ▶ The defect decreases at lower decompression.
- ▶ The defect decreases at lower injection rates.
- ▶ Blisters appear in the injected material.
- ▶ The solidified flow front of a partial filling shows crater-like structures.

CHAPTER 1

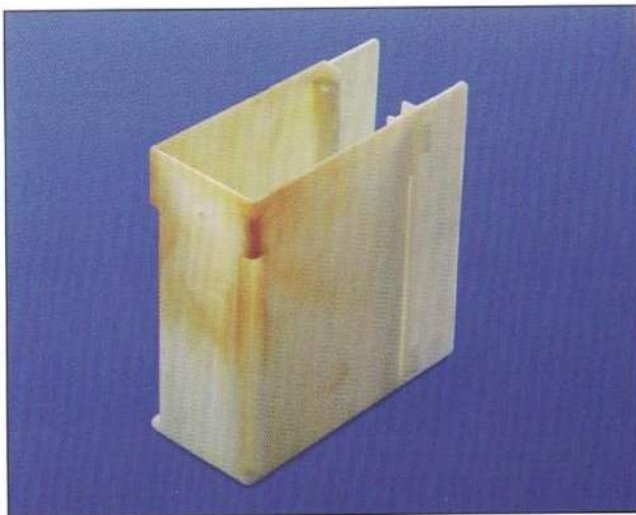
Defect classification
(identification)

1.2.1 Charred streaks

Help on page 55



If excessively high temperatures and/or excessive residence times thermally damage the melt, gaseous decomposition products are produced, which are visible on the surface in the form of brownish or silvery discolorations.



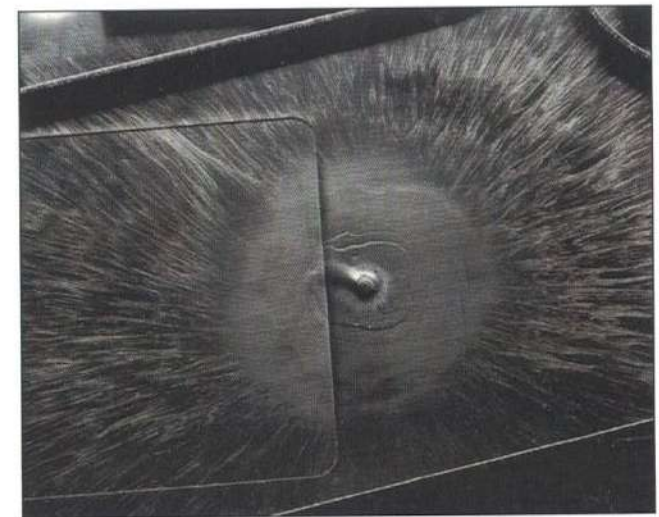
2.1.1

Charred streaks due to an excessive dwell time in the plasticising cylinder



2.1.2

Charred streaks due to high shearing heat in the gate



2.1.3

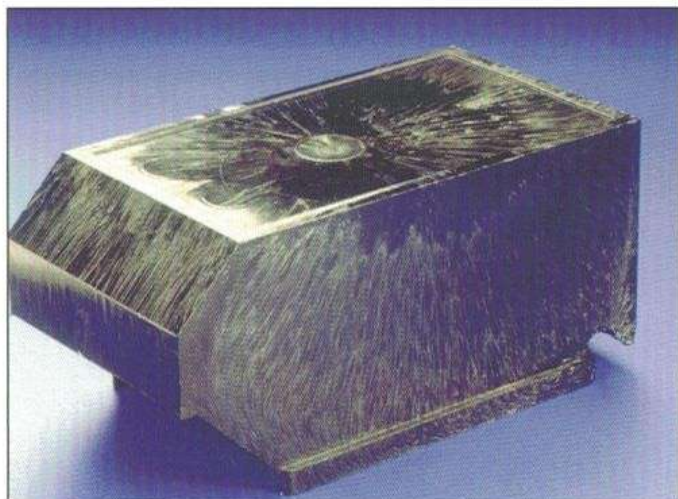
Charred streaks due to an excessive dwell time in the hot runner

1.2.2 Moisture streaks

Help on page 57

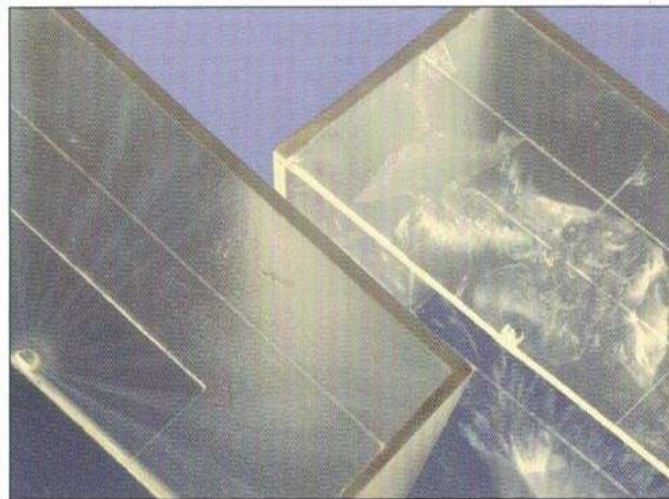


Moisture streaks often appear on the surface of the moulded part in the form of a U-shaped profile. In most cases, the surface around the silvery streak is rough and porous. Moisture streaks, resulting from moisture on the mould surface, appear as large, dull and lamellar areas.



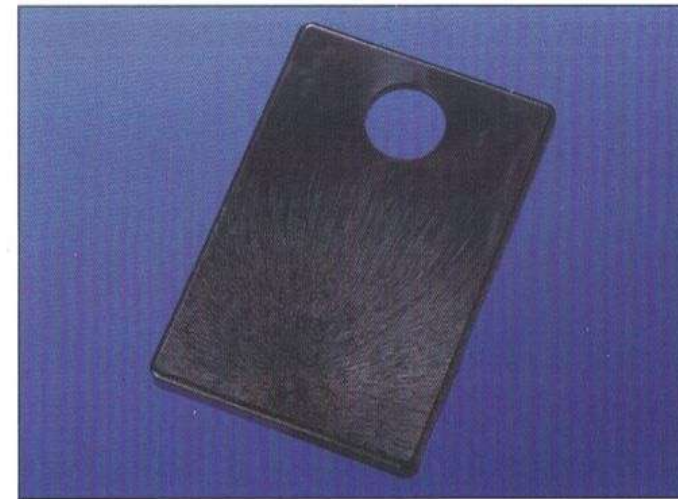
2.2.1

Streaks due to excessive moisture content of the granules



2.2.2

Left: Streaks due to moist granules
Right: Streaks due to moisture on the mould surface



2.2.3

Streaks due to high residual moisture in the granules

CHAPTER 1

Defect classification
(identification)

1.2.3 Air streaks

Help on page 59

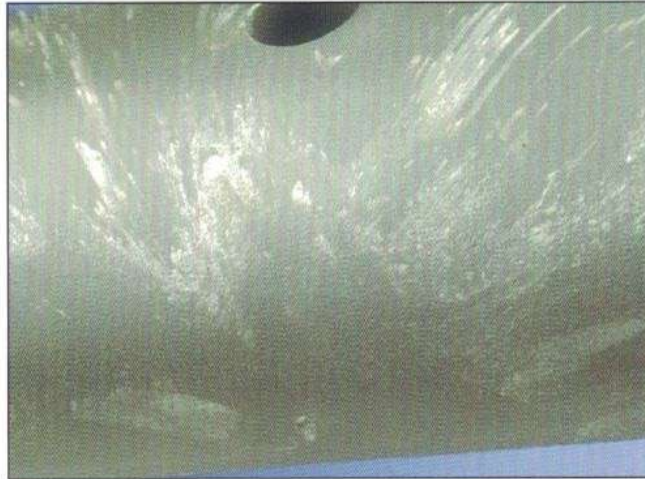


In most cases, air streaks appear in the form of dull, silvery or white streaks and often around domes, ribs and abrupt changes in the wall thickness. Near the gate, lamellar streaks may appear, originating from the gate. Air streaks also appear near letters or depressions.



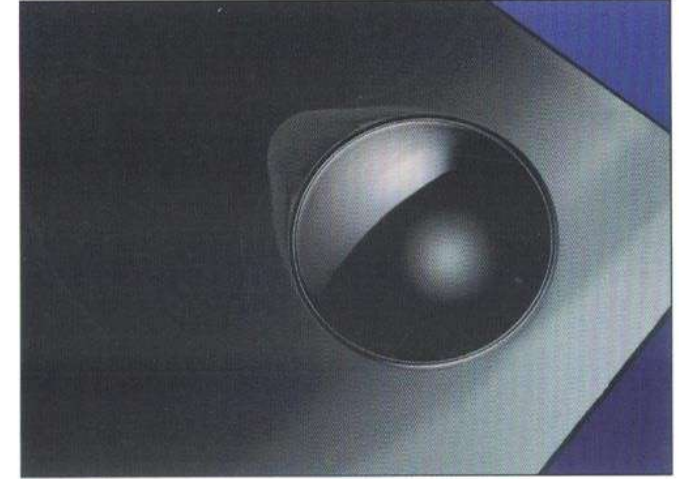
2.3.1

Air streak due to entrapped air



2.3.2

Streak (near the gate) due to air drawn in during decompression



2.3.3

Air streak behind an abrupt change in the wall thickness



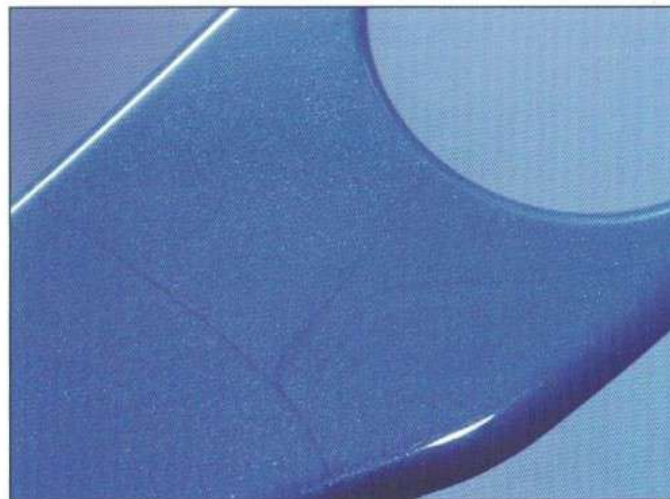
Coloured streaks are the result of an uneven distribution of the used colorant or of a different orientation of the pigments due to the flow process.

Thermal damage and strong deformations on the moulded part, e.g. due to excessive demoulding forces, can also cause colour differences.



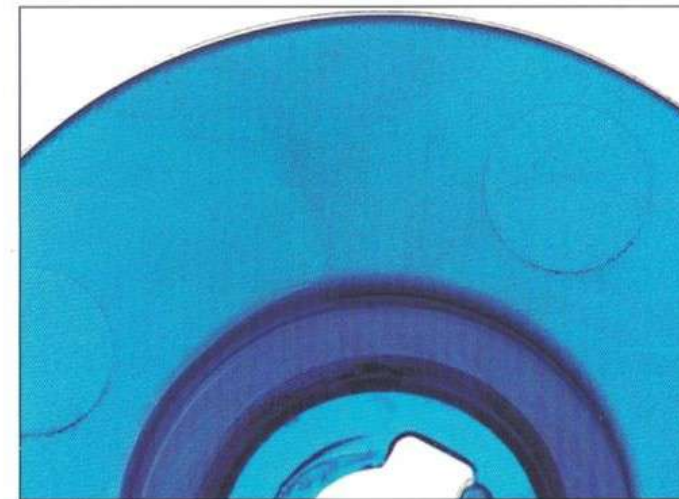
 **2.4.1**

Orientation of metal-effect pigments caused by the flow



 **2.4.2**

Coloured streaks near a weld line



 **2.4.3**

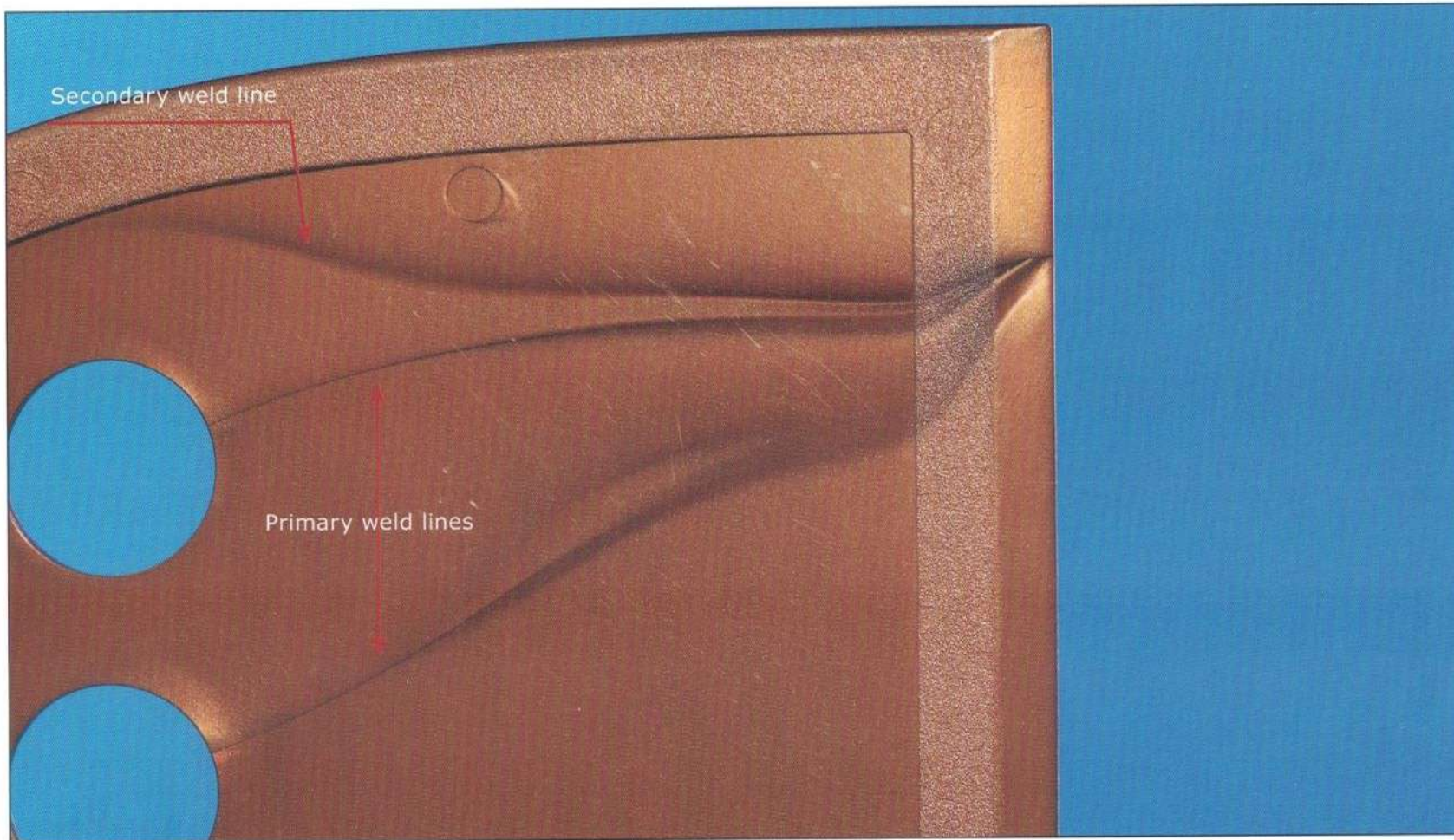
Colour streaks due to an incompatible master batch

CHAPTER 1

Defect classification
(identification)

1.2.4 Coloured streaks

Help on page 61



Note:

Materials with
metallic pigments
are particularly
prone to this
defect.

2.4.4

Streaks around
primary and
secondary weld lines



Mat and rough surfaces can appear when using glass fibre-reinforced materials. The metallically reflecting glass fibres appear on the surface in the form of streaks.



2.5.1

Glass fibre streak due to heavy orientation near the gate



2.5.2

Glass fibre streaks (clearly visible weld line)



2.5.3

Moulded part with rough, silvery surface

CHAPTER 1

Defect classification
(identification)



1.3 Blistering

Help on page 65



Help on classifying (identifying) blisters on moulded parts:

Blisters on moulded parts can be divided into three categories: entrapped air (air bubbles), voids and gaseous entrapments.

Entrapped air is produced during the filling of the moulded part, as air is drawn in by the melt. The air may originate from the plasticising unit or from areas of the mould which are not vented (e.g. ribs or blind holes).

Voids are evacuated cavities, resulting from the impeded shrinkage of the material during the cooling phase.

Gaseous entrapments can also cause blistering. They are usually caused by a chain degradation of the plastic material.

Since the appearance of the defects is very similar, visual differentiation is difficult. The following notes will help to distinguish between the different defects:

Air entrapments (air bubbles):

- ▶ Blisters appear in the injected material.
- ▶ The defect decreases if no decompression is used.
- ▶ Changes to the holding pressure and the holding pressure time have hardly any influence on the size of the cavity.
- ▶ The defect is already visible on partial fillings.

Voids:

- ▶ The defect occurs close to thickwalled areas or material accumulations.
- ▶ Changes concerning the holding pressure or holding pressure time have an influence on the size of the void.
- ▶ The defect mostly occurs during the holding pressure phase.

Gas bubbles (gaseous entrapments):

- ▶ Decreasing the injection rate influences the appearance of the defect.
- ▶ Changing the melt temperature influences the appearance of the defect.
- ▶ Decreasing the residence time influences the appearance of the defect.

1.3.1 Entrapped air (blistering):

Help on page 65



Cavities are visible on or in the moulded part. These are either caused by drawn-in air or mould areas that are not vented and may also occur close to walls.



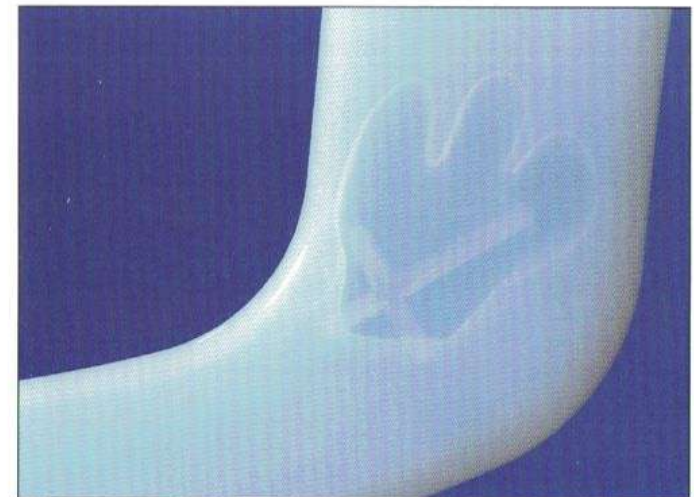
3.1.1

Buckling of the surface due to blistering



3.1.2

Blistering due to injected air



3.1.3

Blisters due to areas of the mould that are not vented

CHAPTER 1

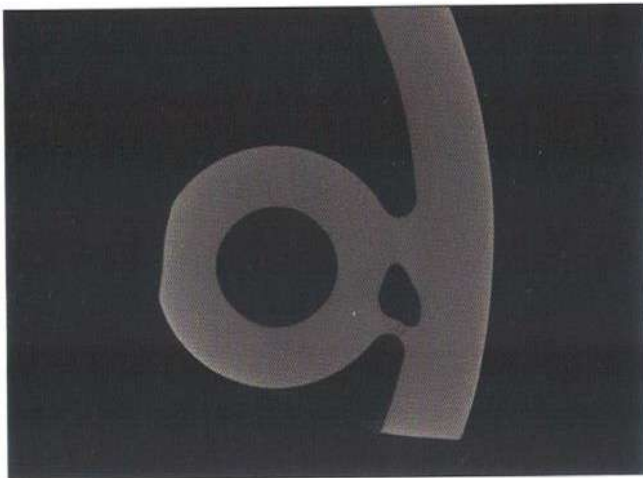
Defect classification
(identification)

1.3.2 Voids

Help on page 67

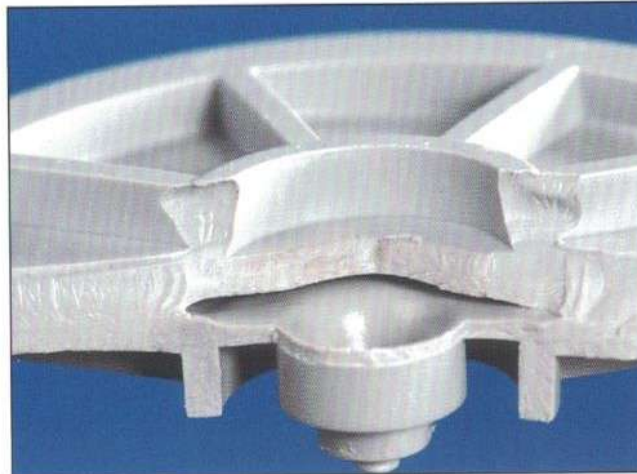


Voids, for example, appear near material accumulations, if the shrinkage can no longer be compensated for.



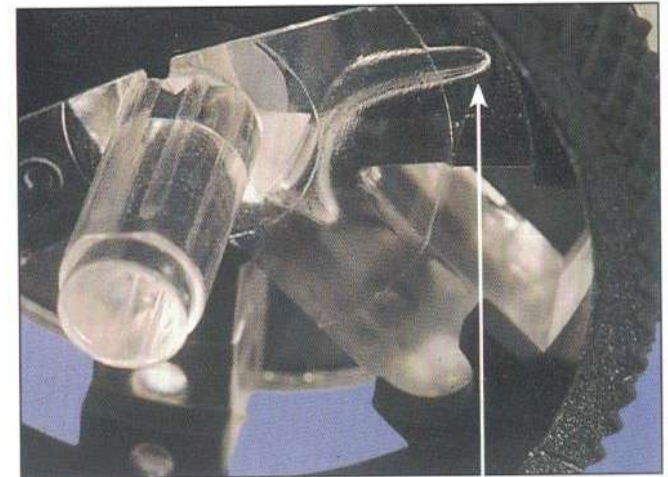
3.2.1

CT image of a void in a thick-walled area of the moulded part



3.2.2

Void near the gate



3.2.3

Void in a light conducting area of the moulded part

1.3.3 Gaseous entrapments

Help on page 69

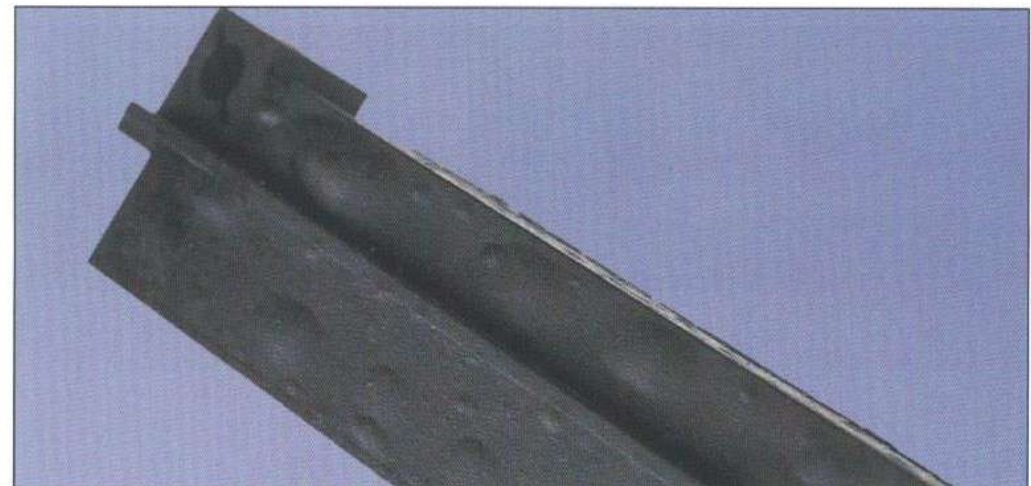


Gaseous bubbles, showing as decomposition products in the moulded part or on the surface, are the result of thermal damage to the material.



 3.3.1

Gaseous entrapments due to depolymerisation (thermal degradation)



 3.3.2

Bubbles on the surface due to thermal material decomposition

CHAPTER 1

Defect classification
(identification)

1.4 Weld line

Help on page 71



The weld line on plastic parts in most cases represents an optical and mechanical weak spot. A notch and/or a colour change may appear. Notches are particularly visible on dark or transparent parts with smooth, highly polished surfaces. Colour changes are particularly visible on parts with metallic pigments.



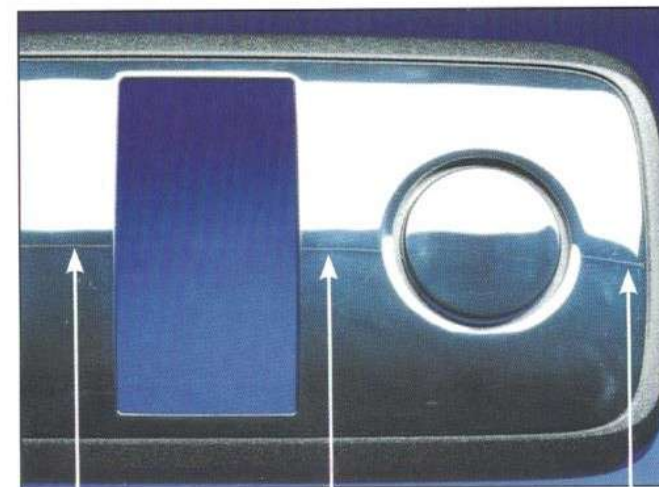
4.1

Screw boss ruptured due to a weld line



4.2

Glass fibre streaks near a weld line



4.3

Visible notch on the top and bottom surfaces of a high-gloss polished moulded part

1.5 Gloss/gloss differences

Help on page 75



As to the evaluation of a moulded part regarding its gloss, two defects may occur. Either the entire moulded part is excessively or insufficiently glossy, or there are gloss differences on the surface of a moulded part. Gloss differences often appear near wall thickness variations on the visible area of the moulded part.



 5.1

Gloss difference in the area of an ejector



 5.2

Gloss differences due to different wall thicknesses



 5.3

Gloss difference due to a wrong PVD process
left: PVD-coated (matt finish)
right: uncoated (glossy finish)

CHAPTER 1

Defect classification
(identification)

1.6 Jetting

Help on page 80



A serpentine - rough or mat - line is visible on the surface of the moulded part. Often jetting causes differences in gloss and colour. In some cases the defect looks similar to the record grooves effect (see page 24).



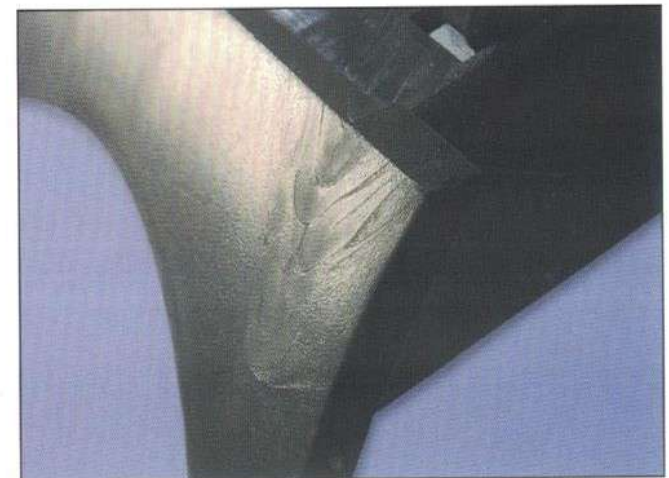
 6.1

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



 6.2

Jetting near the gate



 6.3

Jetting due to inappropriate gating near ribs

1.7 Record grooves effect

Help on page 82

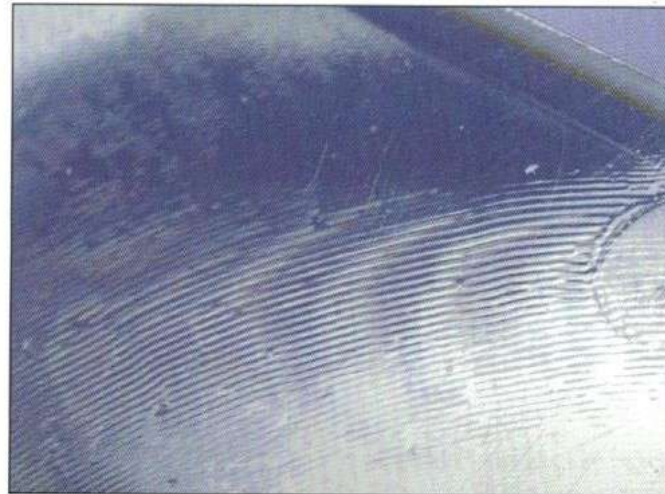


Very fine grooves show up on the surface of the moulded part, resembling those of vinyl records. Concentric rings appear near pinpoint gates, while the markings are parallel towards the end of the flow path and/or behind film gates.



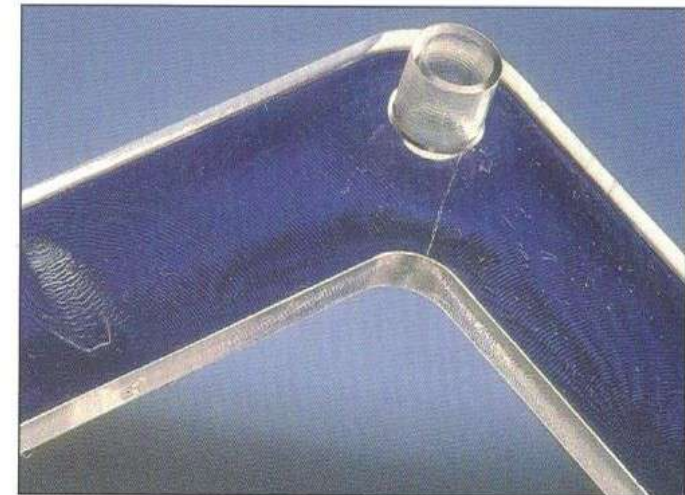
7.1

Concentric record grooves



7.2

Concentric record grooves



7.3

Record grooves originating at the gate, spreading over the entire moulded part

CHAPTER 1

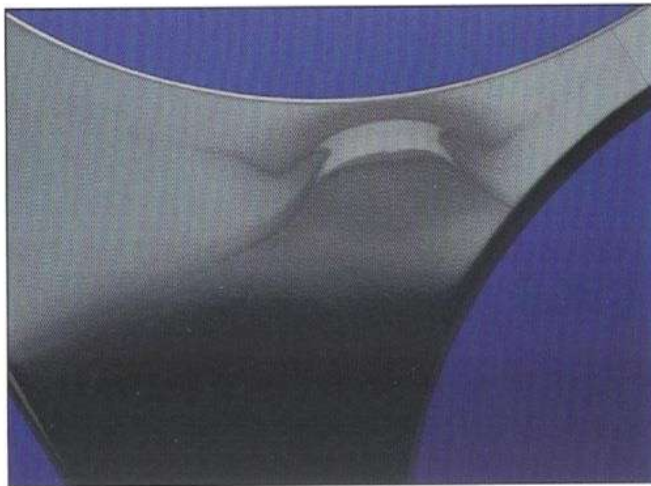
Defect classification
(identification)

1.8 Dull spots near the gate

Help on page 84

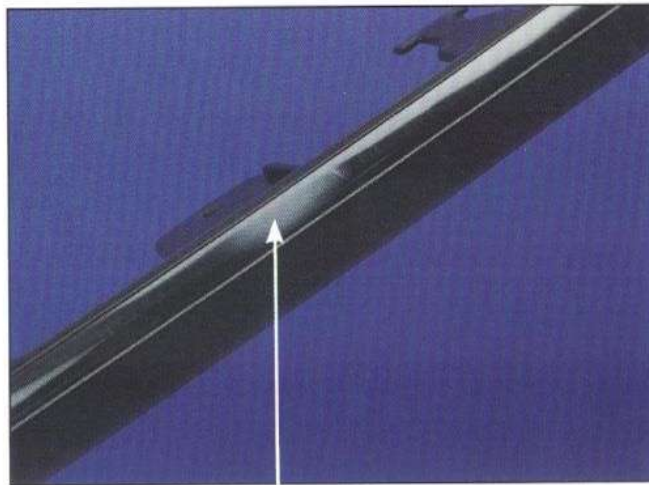


Concentric rings are visible around the gate, which are often so small that only a dull corona is visible.



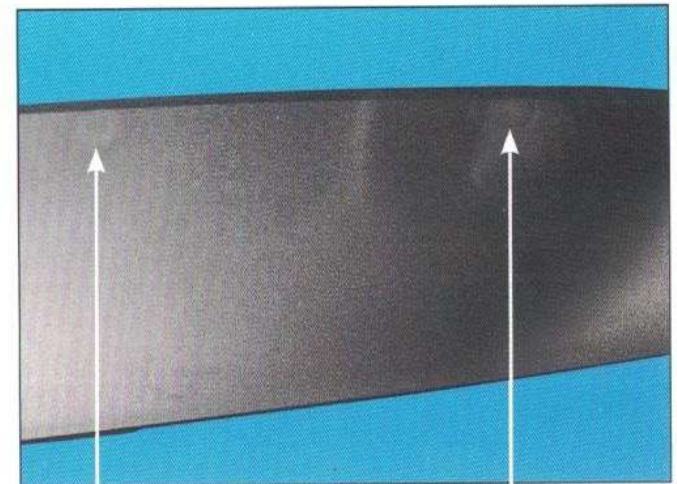
8.1

Dull spot (corona) near the gate



8.2

Dull spot (corona) near the gate



8.3

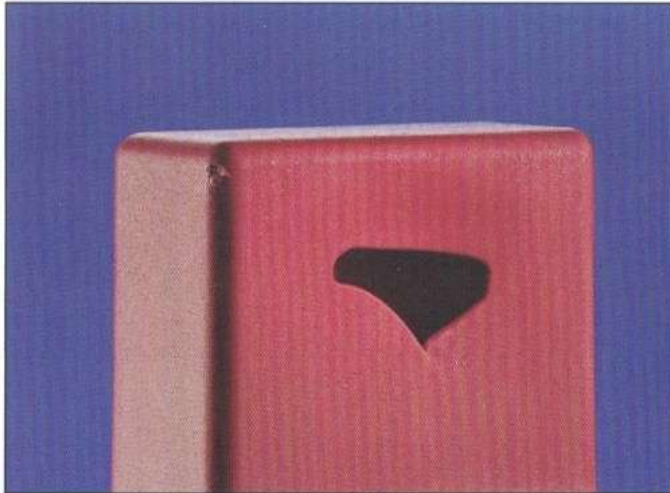
Dull spot (corona) near the gate

1.9 Incompletely filled parts

Help on page 86



Injection moulded parts with incompletely developed outer profiles are described as incompletely filled parts. This kind of defect often appears far away from the gate in the case of long flow distances, or near thin walls (e.g. ribs). This defect can also occur in other areas, due to insufficient mould venting.



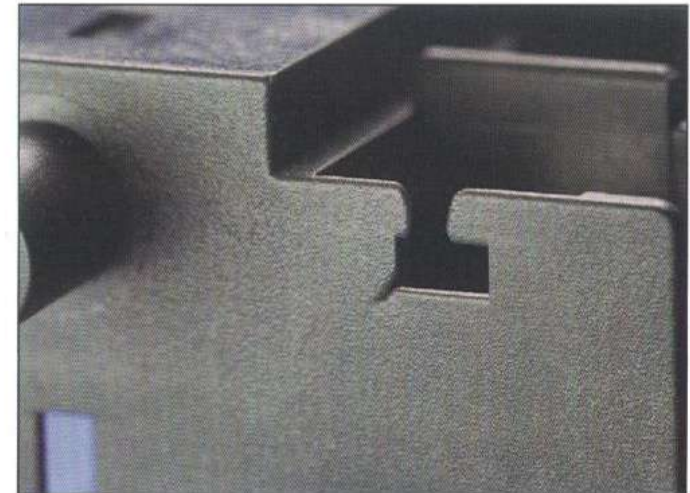
9.1

Incompletely filled part due to entrapped air



9.2

Incompletely filled rib, away from the gate



9.3

Incompletely filled rib, near the gate

CHAPTER 1

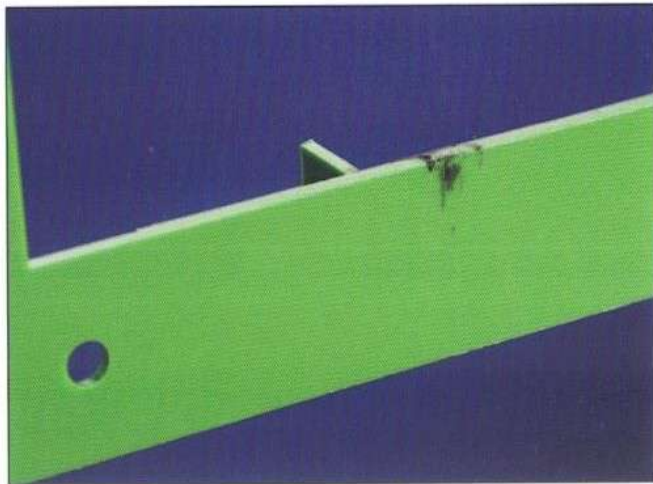
Defect classification
(identification)

1.10 Diesel effect/burn

Help on page 88



Concentrated black spots (burns) are visible on the surface of the moulded part. Often the parts are incompletely filled in these areas or weld lines or other marks appear.



 10.1

Diesel effect due to a lack of venting in the area of a rib



 10.2

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



 10.3

Diesel effect due to merging of several flow fronts (mould filling study)

1.11 Overpacked parts (flashes)

Help on page 90



Flashes often appear near mould parting surfaces, sealing faces, venting channels or ejectors. They look like more or less developed film-like plastic edges. Fine flashes are often not immediately recognised as such. Large area, thick flashes, on the other hand, sometimes stick out several millimetres above the nominal profile.



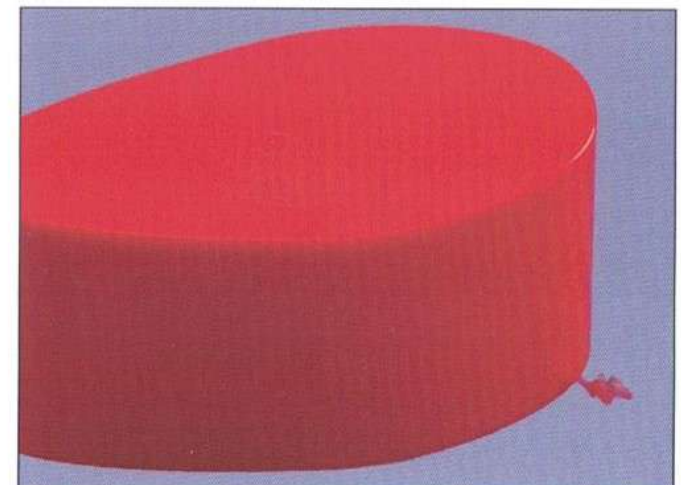
 11.1

Flashing on the parting surface



 11.2

Flash formation on a mould parting surface



 11.3

Flash formation on a venting channel

CHAPTER 1

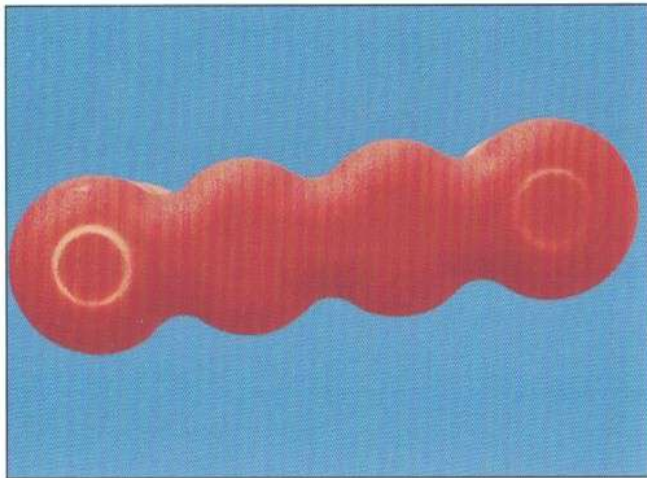
Defect classification
(identification)

1.12 Stress-whitening/stress cracks

Help on page 92



Stress whitening is caused by inner and outer stress (e.g. heavy elongation). The areas exposed to stress turn white. Stress cracks are adjacent cracks, which often take several days or even weeks after production to appear.



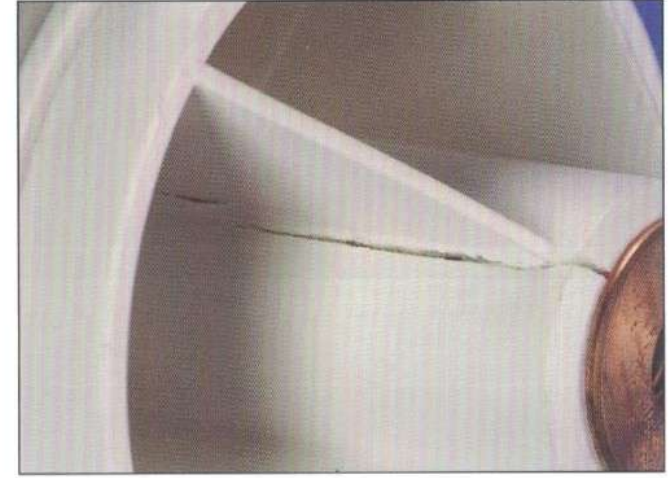
12.1

Stress-whitening around the ejectors
due to stress during demoulding



12.2

Stress cracks on a cover (damage only visible
several weeks after the purchase)



12.3

Stress crack due to notch effect of sharp-edged
metal insert

1.13 Visible ejector marks

Help on page 94



Ejector marks are depressions or elevations and thus constitute abrupt changes in the wall thickness. These wall thickness variations can cause gloss differences and depressions on the visible surface of the moulded part. In the case of high demoulding forces even punctures, caused by the ejector, may occur.



13.1

Damage to the visible surface due to depressed ejectors



13.2

Gloss differences near the ejector



13.3

Damage to the visible surface due to depressed ejectors

CHAPTER 1

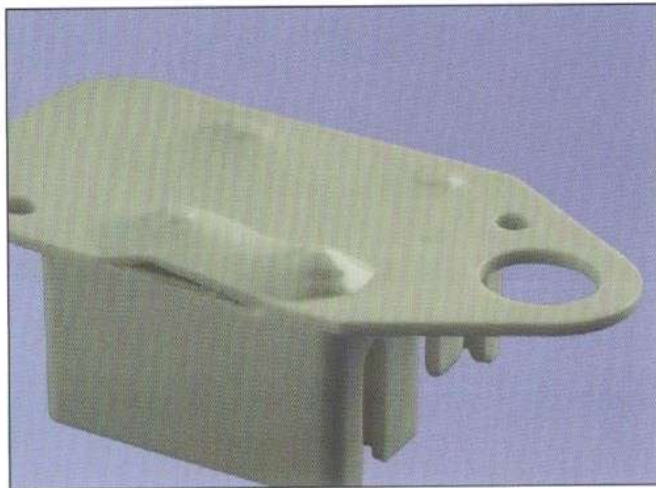
Defect classification
(identification)

1.14 Deformation during demoulding

Help on page 96

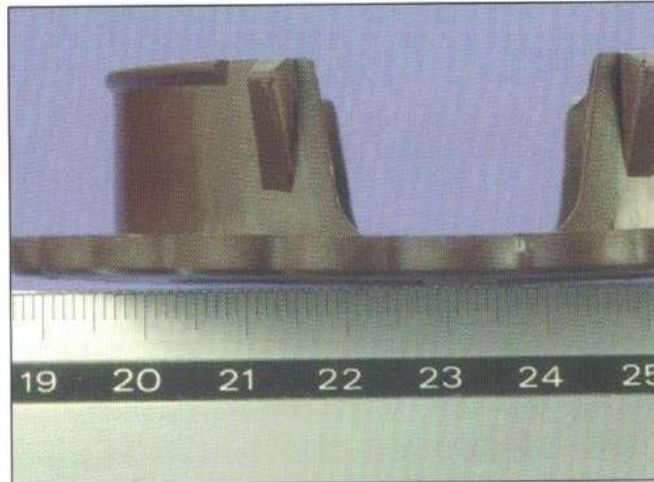


Depending on the degree of damage, there is a classification into cracks, fractures, overstretched areas and deeply depressed or puncturing ejectors. Particularly critical are moulded parts with undercuts, which are demoulded without the use of movable parts (e.g. slides).



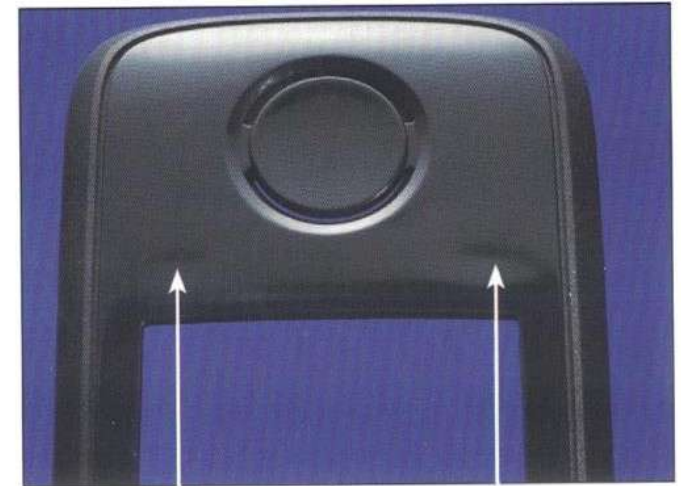
 14.1

Strong part deformation caused by the ejector



 14.2

Deformation due to forced demoulding at an undercut



 14.3

Deformation due to overloaded ribs close to the gate



1.15 Ejection grooves (demoulding grooves)

Help on page 100



Ejection grooves are produced during demoulding and constitute a damage to the surface of the moulded part. They mainly occur on structured surfaces.



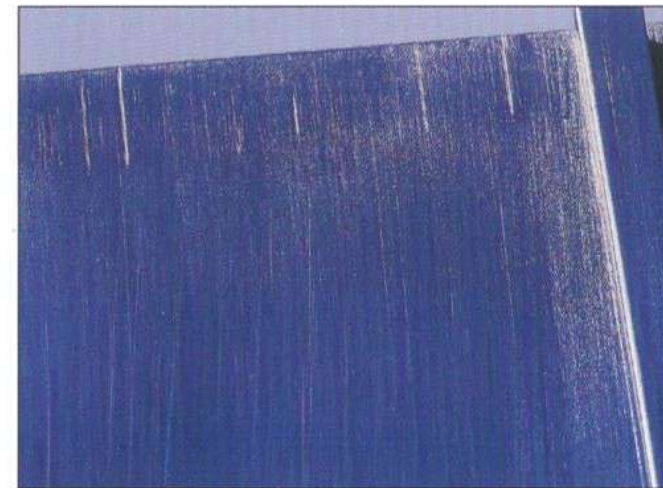
 15.1

Demoulding grooves due to a pressure peak towards the end of the filling of the moulded part



 15.2

Demoulding grooves on a structured surface



 15.3

Demoulding grooves due to a pressure peak towards the end of the filling of the moulded part

CHAPTER 1

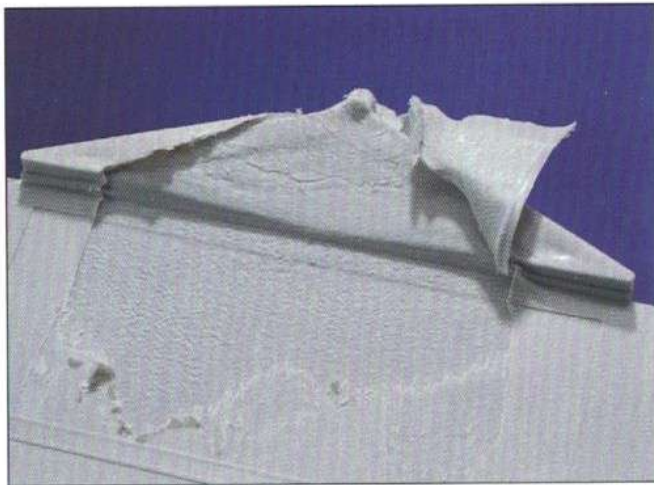
Defect classification
(identification)

1.16 Flaking of the surface layer

Help on page 102



The layers of material are not homogeneously joined together and start flaking. This can occur at the gate or on the moulded part, and can be either large or very small and thin, depending on the intensity.



16.1

Flaking at the film gate and on the moulded part



16.2

Large-area flaking



16.3

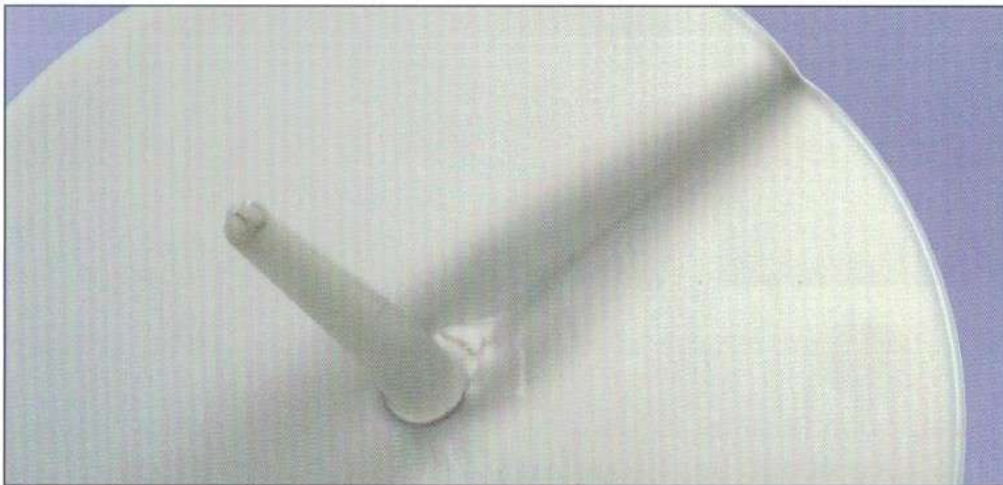
Cosmetics article with flaked off "skin"

1.17 Cold slugs

Help on page 104



Cold melt coming out of the nozzle (or hot-runner) into the mould, can cause marks shaped like comet tails. They can appear near the gate or spread all over the moulded part. Cold slugs can also cause weld lines, if jamming a runner and forcing the melt to part.



 17.1

Weld line behind a cold slug



 17.2

Cold slug (near gate)

CHAPTER 1

Defect classification
(identification)

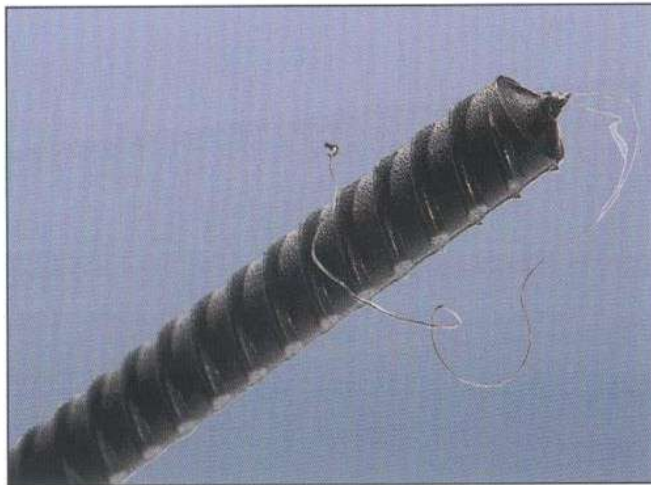
1.18 Filamentation

Help on page 106



Filamentation can occur in two areas of the injection mould. In the case of direct hot runner gating, a thin filament may stick to the moulded part around the gate area. The shape may range from a short pointed cone to a filament of several centimetres.

A long filament may also be produced between the hot runner system and the machine nozzle, which will be elongated when opening the mould.



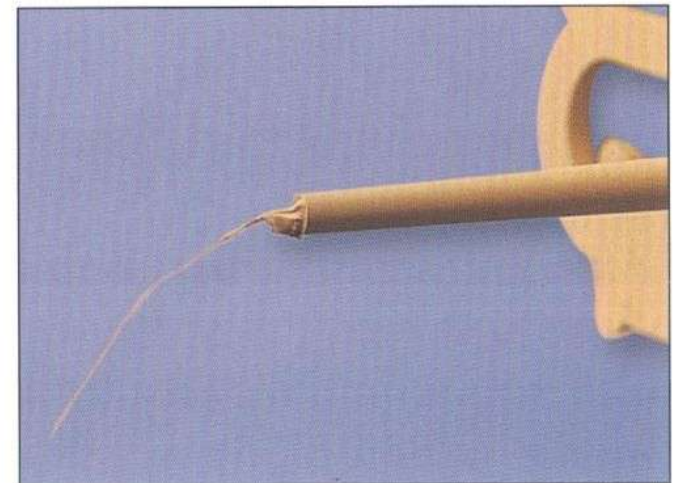
18.1

Long filament near a hot runner gate



18.2

Short cone with short filamentation in the area of a hot runner gate



18.3

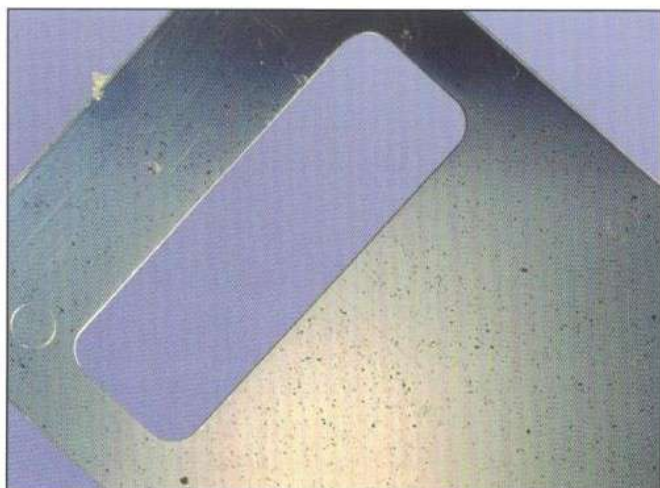
Long filamentation at a cold runner gating system near the machine nozzle

1.19 Dark spots

Help on page 108



Black or dark spots appear on the surface, caused by wear, thermal damage or dirt.



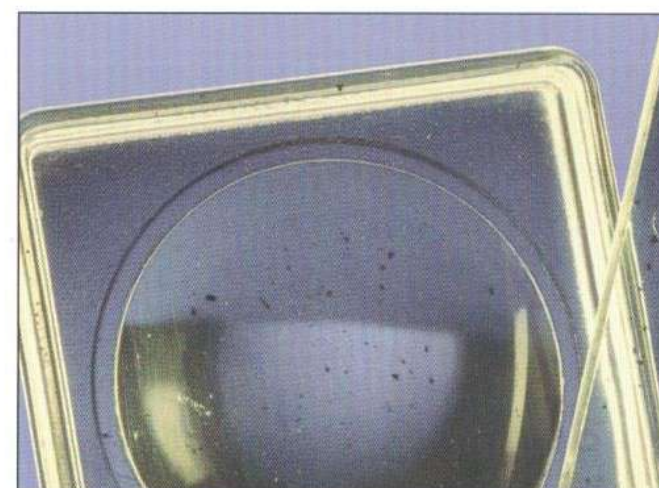
 19.1

Dark spots due to thermal damage



 19.2

Dark spots after changing the material



 19.3

Dark spots due to thermal damage

CHAPTER 1

Defect classification
(identification)

1.20 Plate-out

Help on page 111



Low-molecular weight separations from the plastic melt can cause plate-out after just a few thousand cycles on the mould surfaces in contact with the plastic, mostly in areas close to the gate or the venting. This plate-out prevents an exact reproduction on the cavity surface. The results are, for example,

topographic defects on the surface of the moulded part or gloss differences.



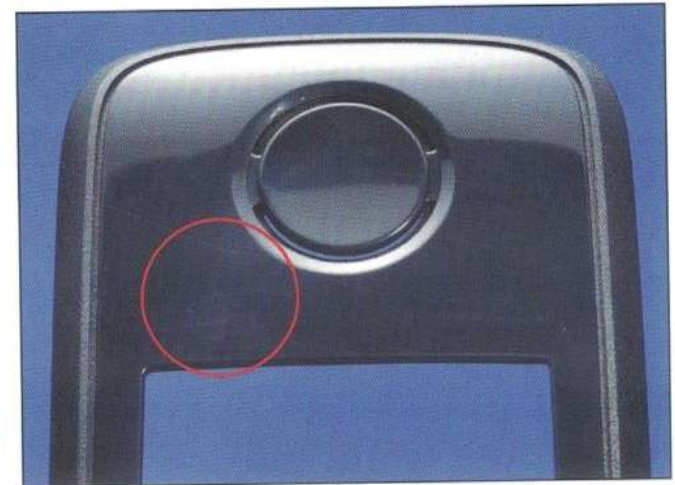
20.1

Rough surface due to plate-out in the mould



20.2

Plate-out, near the gate



20.3

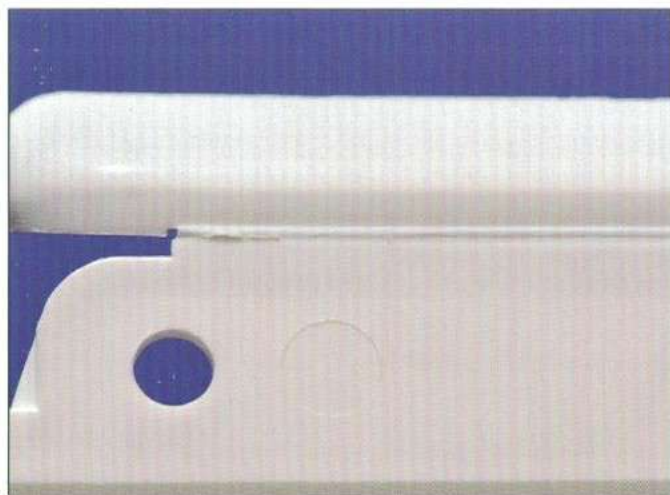
Partial gloss differences due to plate-out in the mould

1.21 Broken/ragged film hinge

Help on page 114



The failure of film hinges is mainly caused by overstressed plastic. The film hinge may break partially or completely. Over-stressing can also result in stress whitening.



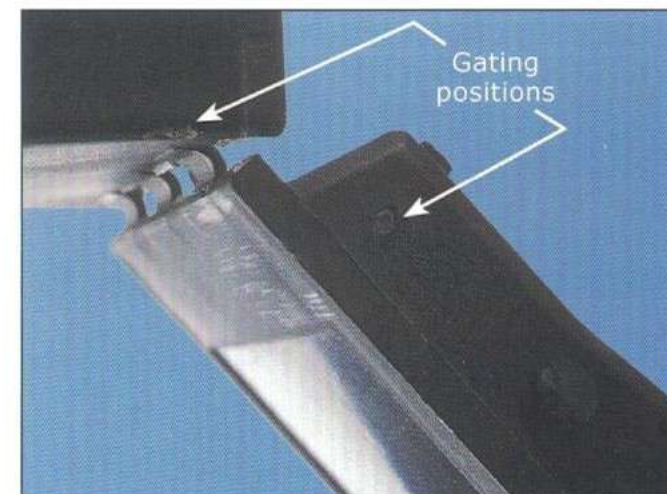
 21.1

Partial split of a film hinge



 21.2

Melt standstill inside film gate (broken film gate)



 21.3

Weld line in the film hinge, caused by the gating positions

CHAPTER 1

Defect classification
(identification)

1.22 Warpage

Help on page 116

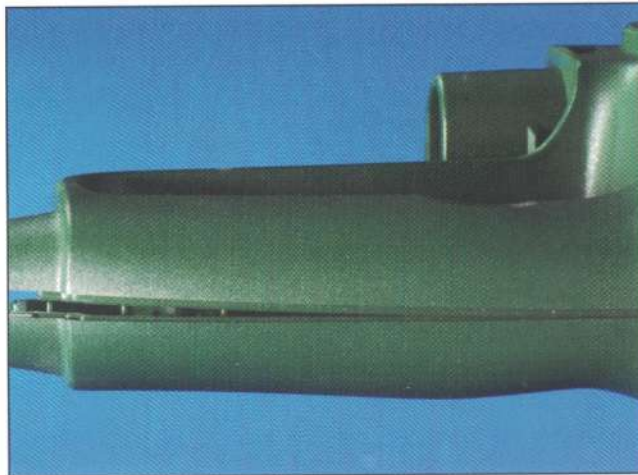


Warpage is the deviation of a moulded part from its required shape. Warpage is always the result of shrinkage differences, with the exception of distortions caused by deformation during demoulding.



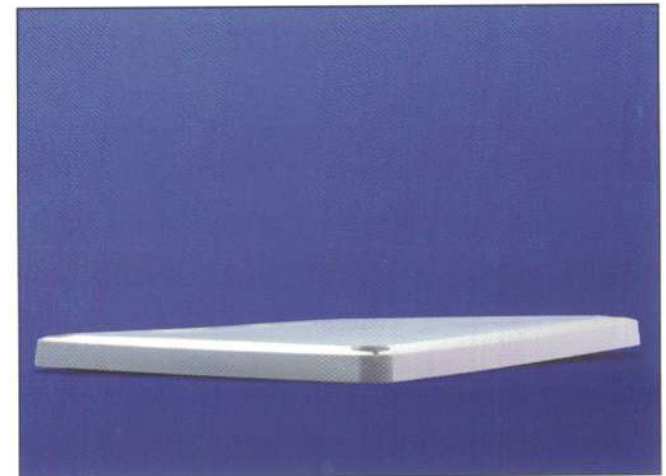
22.1

Warpage on box-shaped parts



22.2

Warpage due to glass fibre orientation



22.3

Warpage of a cover

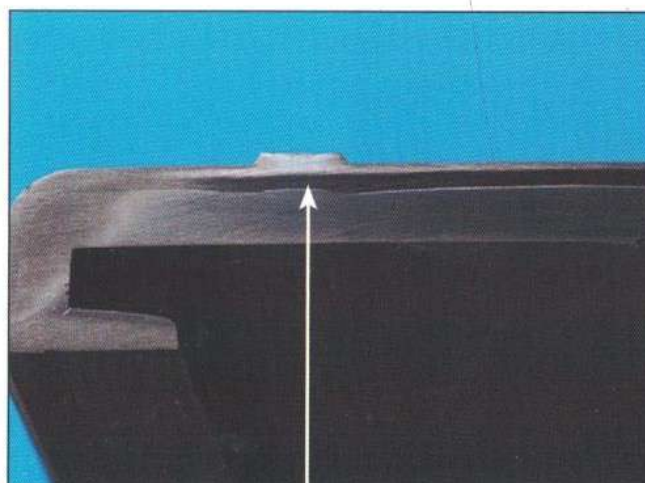


The quality of plastic parts produced with multi-component technology depends not only on the quality of the individual components, but also on their mutual influence on each other, such as warpage, adhesion and erosion in the gate area.



 23.1

Cold flow of the soft component due to the holding pressure



 23.2

Incipient fusion of the first component



 23.3

Poor adhesion

CHAPTER 1

Defect classification
(identification)

1.24 Tiger lines

Help on page 129



Tiger lines are alternating gloss differences (dull – shiny) on moulded parts, which generate differences in the surface roughness. These flow marks are visible on both sides of the surface of the moulded parts. They are characterized by a notable regularity and usually spread over the entire surface of the component in the form of concentric rings, starting at the gate. Another peculiarity of this defect is that an area on one side of the moulded part is shiny, while the area on the opposite side appears dull.



 24.1

Alternating or pulsating melt centre



General notes concerning thermoplastic foam injection moulding (TSG)

The main arguments in favour of thermoplastic foam injection moulding are the reduced weight, material, cycle time, and warpage, as well as a shrinkage-compensating function. During this process, a thermoplastic melt is loaded with a blowing agent and injected into the mould cavity. The most important disadvantage of the various foaming techniques is the formation of streaks. This is why visible parts may be problematic or at least require additional measures for concealing the surface defects.

As a matter of principle, a distinction can be made between chemical and physical foaming. For chemical foaming, certain active agents are used. They decompose under the influence of heat and release a gas in the process. In this context, heat-releasing (exothermic) and heat-absorbing (endothermic) agents are distinguished. The agents are added to the base poly-

mer in the form of master batches, powder, or liquids, and they are plasticised together with the polymer. The blowing agents must be precisely adapted to the base polymer in order to achieve an optimal decomposition reaction. For physical foaming, a gas (N , CO_2) is added to the melt under high pressure during or after the plasticisation. During this process, high gas pressures are realised, making the process particularly suitable for „thin-walled“ parts. In addition, the foaming process is independent of the processing temperature and it can be realised without any reaction residues. A disadvantage compared to chemical foaming is that it requires more sophisticated and rather specific equipment.

Both processes present special challenges to the systems, equipment, and moulds, and the fulfilment of these requirements is a prerequisite for the production of high-quality products.

Since the gas should not expand until the melt is in the cavity, the melt must be kept

under pressure beforehand. This is why the following points should be taken into consideration in view of the equipment:

- ▶ Provision of a shut-off nozzle in the unit or hot runner (the correct closing and opening times must be taken into consideration).
- ▶ Screw position control during all of the process phases (also in the event of malfunctions).

The following must be taken into consideration concerning the part and mould design:

- ▶ Filling from thin to thick.
- ▶ Even wall thickness transitions.
- ▶ Excessive wall thickness differences should be avoided.
- ▶ In view of the surface quality and mechanical aspects, the weld line areas are more critical than in the case of compact injection moulding.
- ▶ The gate runners must be smaller than for compact injection moulding.
- ▶ Natural balancing of the runner in the case of multiple cavities should be

1.25 Defects of thermoplastic foam injection moulded parts

Help on page 132



- avoided (preferred use of hot runners).
- ▶ Optimisation of the venting system (perhaps a variable design).

The process control also requires fundamental changes when gas-loaded melts are processed. The following section provides some basic notes on the fundamental definition of the process parameters for thermoplastic foam injection moulding (TSG). These notes and the following representations for the classification and elimination of defects refer solely to the use of chemical blowing agents in the form of master batches.

Melt temperature

- ▶ Orientation based on the decomposition temperature of the blowing agents (should be reached in the compression zone).
- ▶ „Cold“ configuration of the feed sec-

tion in order to avoid the premature reaction and softening of the carrier material (master batch).

- ▶ It may be possible to reduce the melt temperature by reducing the viscosity.

Mould temperature control

- ▶ It could be beneficial to reduce the temperatures in the case of exothermic materials.
- ▶ Ensure a homogeneous mould wall temperature (hot spots must be avoided).

Plasticisation

- ▶ Deactivate the decompression.
- ▶ Select a (permanent) back pressure that is higher than the gas pressure.
- ▶ The dosing time should end when the residual cooling time ends.
- ▶ In general, the blowing agent is not dried together with the other material.

Injection phase

- ▶ Adjust a higher melt front velocity (keep the gas under pressure).
- ▶ Time-lagged opening of the shut-off nozzle during the injection should be considered.
- ▶ Keep the melt cushion small.

Holding pressure phase

- ▶ If possible, do not use any holding pressure (the pressure can prevent foaming)
=> complete deactivation so that the nozzle closes.
- ▶ Influence on the peripheral layer by minimal holding pressure.

Residual cooling time

- ▶ It could be beneficial to increase it in the case of exothermic materials.
- ▶ Reduced warpage may enable earlier demoulding.



25.1

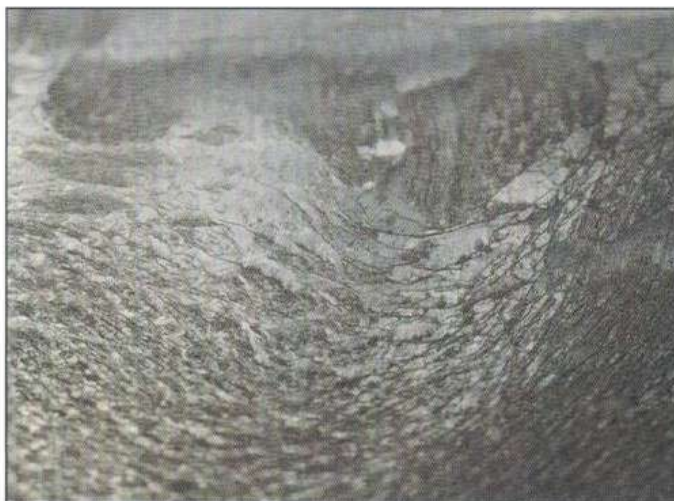
Silvery streaks on the entire surface



Foamed parts often have a slightly streaky surface. However, highly inhomogeneous streaks and massive gas hooks may form on the surface as a function of the flow path. These defects can be influenced by the process parameters.

25.2

Folded surface



Particularly at high gas pressures and injection rates as well as at abrupt wall thickness changes, a slab-like surface structure and folds at the end of the flow path may result.

25.3

Formation of streaks due to venting problems at the end of the flow path



During thermoplastic foam injection moulding (TSG), streaks may occur in the venting areas. The defect can be caused by an insufficient number of venting channels and/or excessive clamping forces as well as in the opposite case.

CHAPTER 1

Defect classification
(identification)

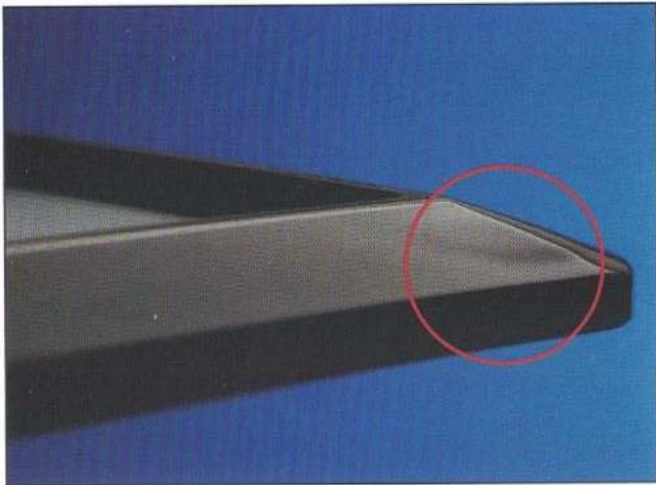
1.25 Defects of thermoplastic foam injection moulded parts

Help on page 132



25.4

Sink mark in an area with an abrupt change in wall thickness *1



Despite the use of blowing agents, partial sink marks may be produced in the area of abrupt wall thickness changes (thick to thin).



25.5

Formation of pimples/bumps *1



In chemically foamed parts, pimples or bumps may form due to the reaction of blowing agent residues on the part surface with water vapour.



25.6

Air hooks *1



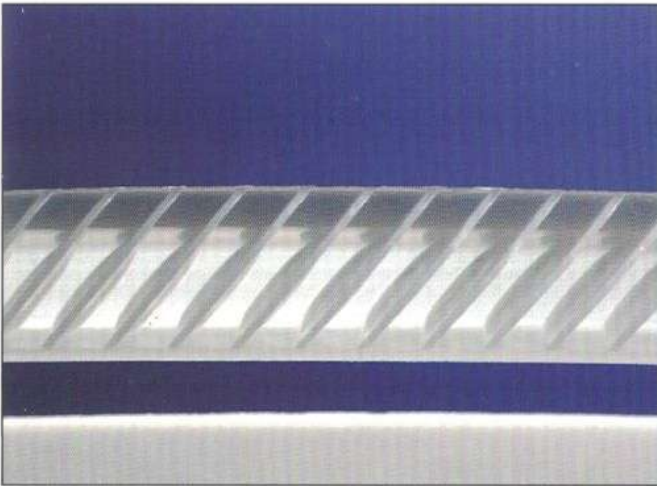
Air hooks can be caused by expansion in the space in front of the screw or in the hot runner.

*1 Thermoplastic foam injection moulded parts (TSG parts) in combination with dynamic temperature control (heating and cooling)



25.7

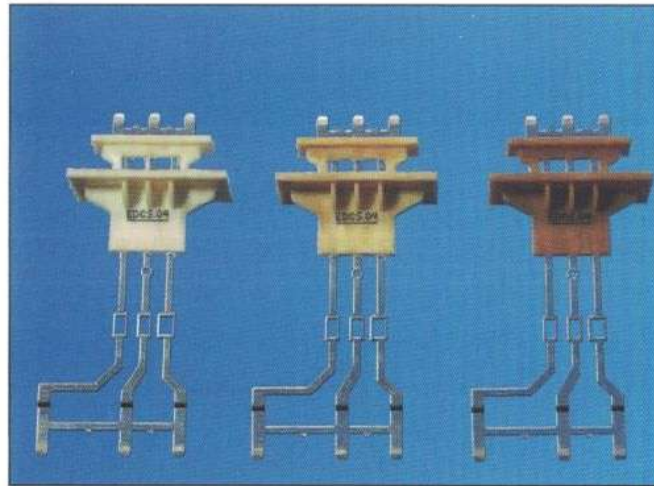
Inflated areas at the rib base



Due to the gas load of the melt, foam injection-moulded parts may show partial inflation/bulging. In view of the potential cause of the defect, it should be determined as to whether the defect occurs directly after demoulding or after a certain waiting time.

25.8

Colour changes



Particularly in the case of exothermic blowing agents, the moulded parts may show a brownish change in colour over a certain production period. This colour change is caused by a thermal degradation of the material.

25.9

Plate-out on a mould surface



When chemical blowing agents are used, deposits and/or corrosion may form on the mould surface. They depend on the type of active agent and base polymer and are due to the resulting reaction products.

CHAPTER 1

Defect classification
(identification)

1.26 Unevenly filled parts/flow anomalies

Help on page 140



The uneven filling of parts may be due to several different causes and result in different types of defects. In many cases, the defects are present in the form of atypical melt front courses that may lead, for example, to an uneven filling of multiple cavities or to air entrapments or secondary weld lines. These flow anomalies can often be attributed to relatively intense or long-lasting material shearing. As mentioned above, this advancing can occur in one cavity or between several cavities in the form

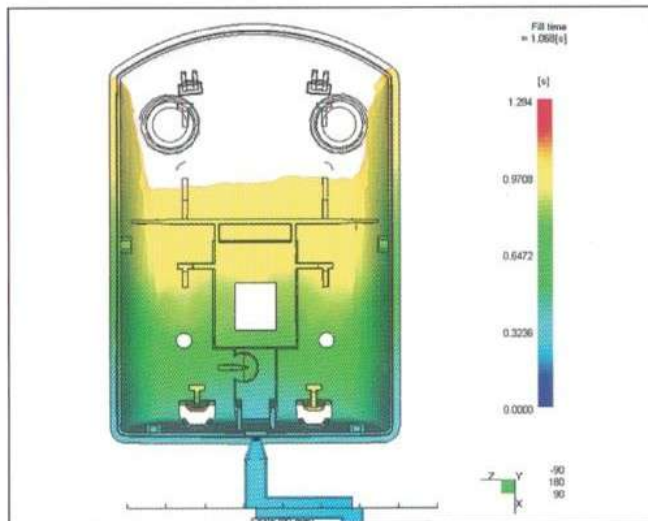
of a balancing problem. In this case, natural balancing (flow paths of equal length and cross-section) of the hot and cold runner system is insufficient.

With some variants, naturally balanced cold runners can lead to uneven filling between the individual cavities even though the flow paths have the same length in the individual runners (figure 26.1 and 26.2).



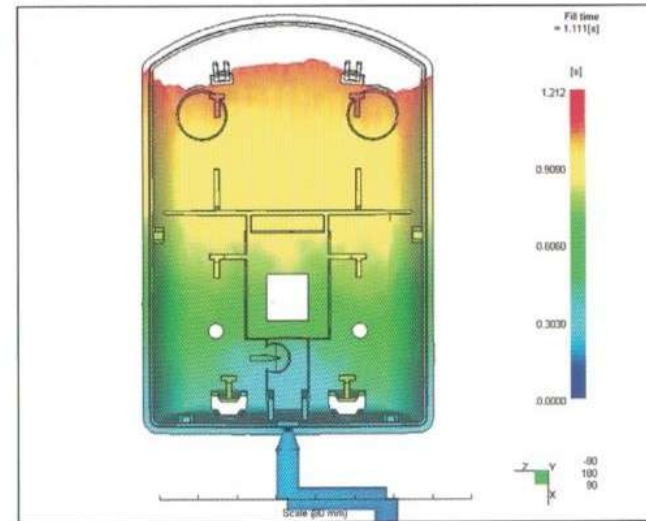
26.1

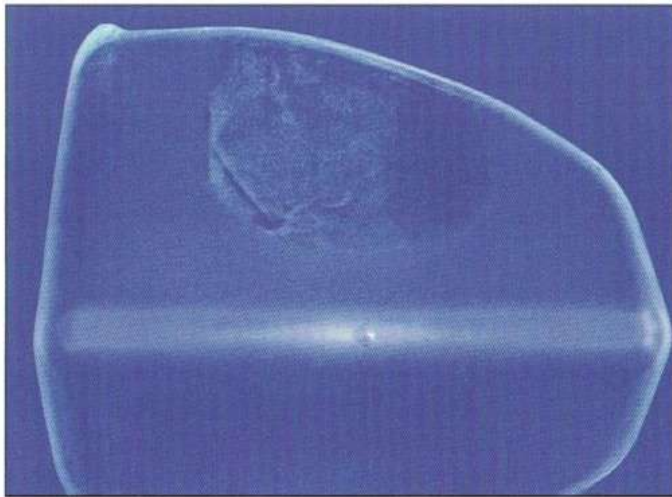
Atypical advancing
of the flow front (low
melt temperature,
fast injection)



26.2

Typical flow front
profile (high melt
temperature, slow
injection)





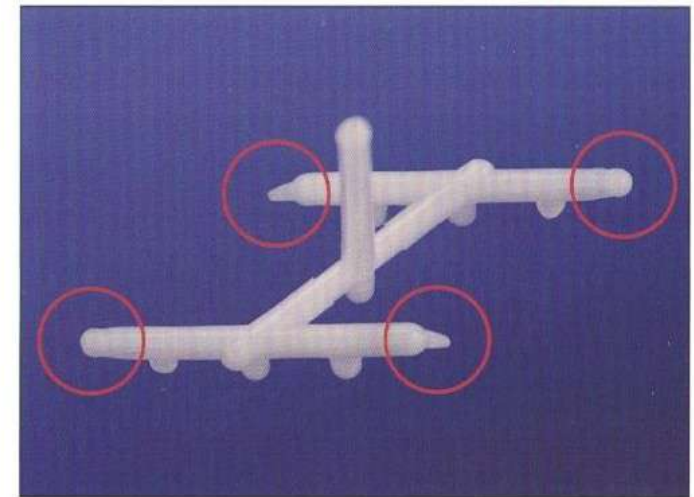
 26.3

Unsymmetrical filling due to a torpedo nozzle



 26.4

Uneven filling due to core displacement (filling study)



 26.5

Uneven filling of a naturally balanced H-distributor (partial filling)

1.27 Dimensional and weight variations

Help on page 145



Dimensional and weight variations are often caused by an instable injection moulding process. Such defects can be attributed to numerous factors, but are usually caused by inappropriate parameter settings (such as insufficient holding pressure or time) or by worn plasticising units, resulting in melt cushion variations.

The following influencing factors are relevant:

- ▶ Melt cushion variation
- ▶ Holding pressure
- ▶ Mould wall temperature

Melt cushion variation

Melt cushion variations can be caused by:

- ▶ Wear on the non-return valve
- ▶ Wear on the plasticising cylinder (erosion on the cylinder wall)
- ▶ Closing speed of the non-return valve

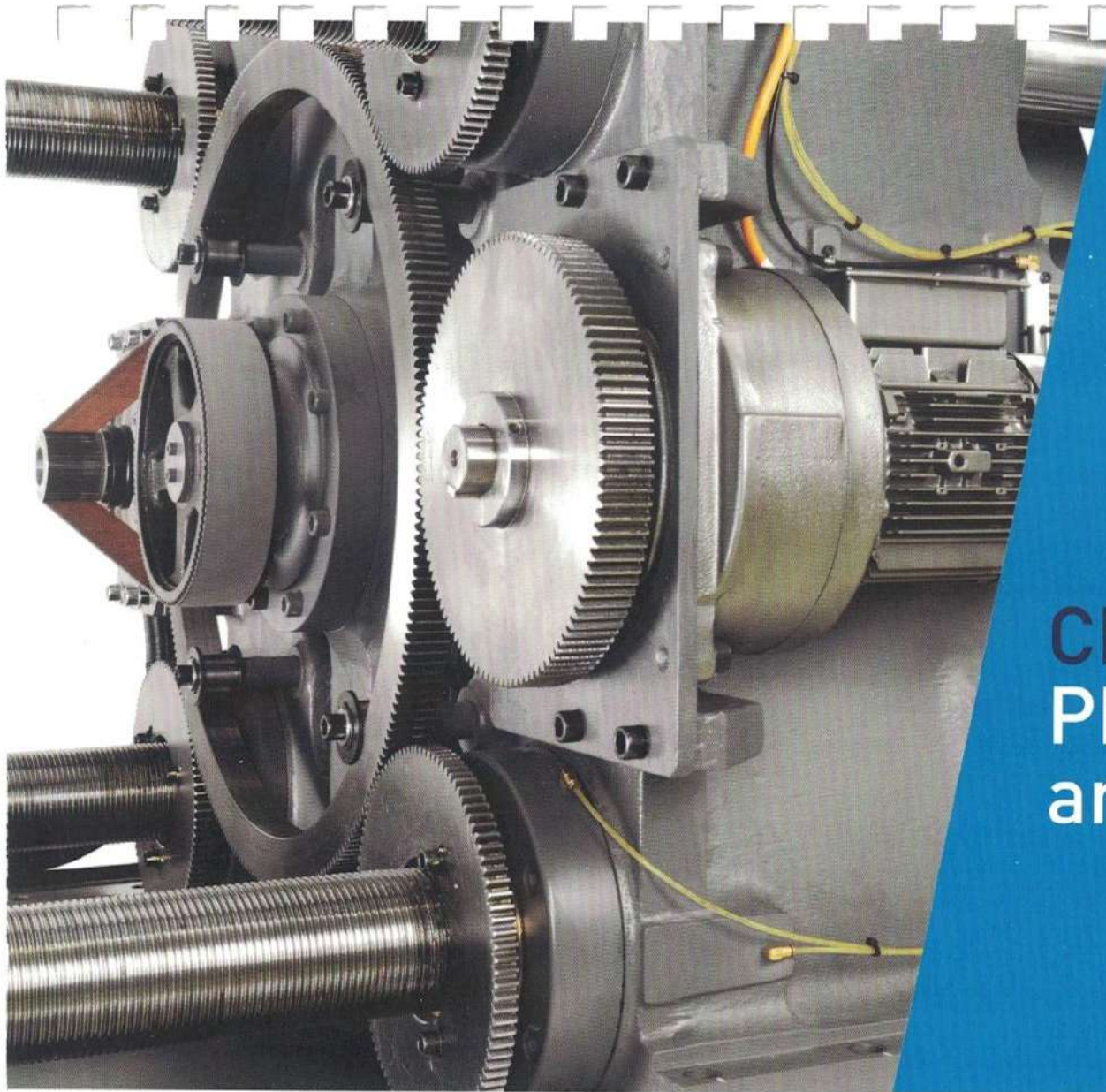
Holding pressure

The holding pressure effectively influences the shrinkage and thus the dimensional and weight stability of the moulded part. The determination of the effective holding pressure time and the gate sealing point are thus crucial to the quality of the moulded part. The gate sealing point depends on the geometry of the moulded part and of the gate and describes the point in time at which material stops entering the cavity to compensate for shrinkage.

Mould wall temperature

The mould wall temperature also has a substantial influence on the total shrinkage of the moulded part. The dimensional stability depends on the selected cooling time and the resulting moulding shrinkage and post shrinkage. In principle, higher moulding shrinkage provides for better dimensional stability (see section 2, defect 22, warpage).

High moulding shrinkage inside the mould under defined boundary conditions reduces dimensional variations.



Chapter 2

Physical causes and remedies

CHAPTER 2

Physical causes
and remedies



Table of contents Chapter

No.	Topic	Page		No.	Topic	Page	
		Cause	Remedy			Cause	Remedy
1.	Sink marks	7	53	11.	Overpacked parts (flashes)	28	90
2.	Streaks	8	55	12.	Stress-whitening/stress cracks	29	92
2.1	Charred streaks	11	55	13.	Visible ejector marks	30	94
2.2	Moisture streaks	12	57	14.	Deformation during demoulding	31	96
2.3	Air streaks	13	59	15.	Ejection grooves	32	100
2.4	Coloured streaks	14	61	16.	Flaking of the surface layer	33	102
2.5	Glass fibre streaks	16	63	17.	Cold slugs	34	104
3.	Blistering	17	65	18.	Filamentation	35	106
3.1	Air entrapments	18	66	19.	Dark spots	36	108
3.2	Voids	19	67	20.	Plate-out	37	111
3.3	Gaseous entrapments	20	69	21.	Broken/ragged film hinge	38	114
4.	Weld line	21	71	22.	Warping	39	116
5.	Gloss/gloss differences	22	75	23.	Multi-component technology	40	124
6.	Jetting	23	80	24.	Tiger lines	41	129
7.	Record grooves effect	24	82	25.	Defects of thermoplastic foam injection moulded parts	42	132
8.	Dull spots near the gate	25	84	26.	Unevenly filled parts/flow anomalies	47	140
9.	Incompletely filled parts	26	86	27.	Dimensional and weight variations	49	145
10.	Diesel effect/burn	27	88				

Flow diagrams for the elimination of defects

The elimination of surface defects may be extremely difficult, due to the diverse physical causes. To make it easier for injection moulders, this chapter contains flow diagrams for the systematic elimination of defects. It is the objective of this procedure to achieve the desired quality by varying the process parameters.

For the application, please note that for each occurring surface defect, the diagram will branch out according to the questions asked. To find the solution to a problem, only one parameter should be changed at a time, to prevent mutual interaction. Several cycles should then be completed to ensure stationary conditions. In some cases, the direction in which the process

parameters should be changed is impossible to specify, but tendencies (+) or (-) are given. If these variations do not bring about any visible improvements, the investigation must be started again and the changes must be applied one after the other. Of course, these diagrams - although taking practical conditions into account - can only serve as suggestions and cannot cover all eventualities that may occur during the process. In addition, they can easily be adapted to special conditions, or completed with your own knowledge. The diagrams shall also help to decide whether the defect can be eliminated at all by changing the machine settings, or whether the mould or the part has to be changed.

CHAPTER 2

Physical causes
and remedies

2.1 Sink marks

Defect described on page 7



Physical cause

Sink marks occur if, during the cooling process, the thermal contraction (shrinkage) of the plastic can no longer be compensated for in certain areas. If the outside walls of the moulded part are not stable enough, due to insufficient cooling, the outer layer is drawn inside by cooling stresses.

There are three fundamental cases:

- ▶ The solidification is too slow
- ▶ The effective holding pressure time is too short
- ▶ There is not enough holding pressure transfer, because flow resistances in the mould are too high or because there are narrow passages in the moulded part or the gating system

Note:

For optimum holding pressure transfer the moulded part should be gated to the largest crosssection. If this is not possible, a flow promoter between the gate and the melt accumulations may improve the holding pressure transfer. In order to avoid premature solidification of the sprue and gate system, sufficient dimensioning is necessary. After eliminating the sink marks, the moulded part must be checked for voids.

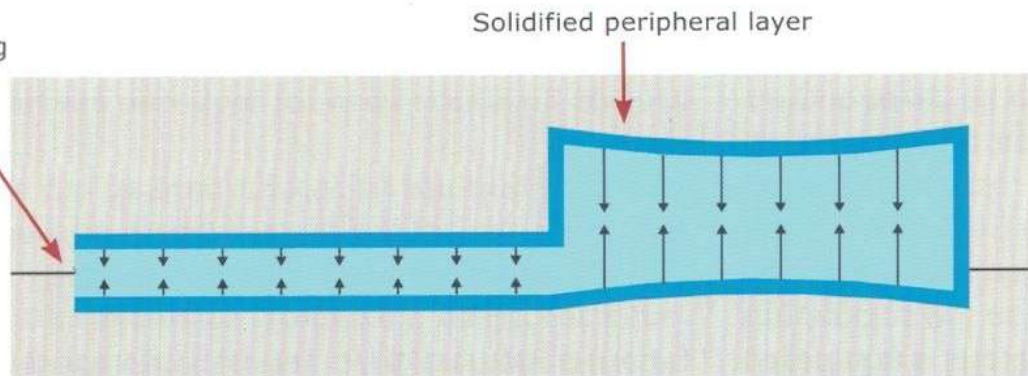
High-gloss surfaces often have a bumpy, rippled surface at the end of the flow path. This defect resembles a sink mark. However, it is caused by insufficient venting.

Disadvantageous gating



1.1

Moulded part with
sink marks (gating at
the thin wall)





Investigation

Check and/or change machine settings, change mould or moulding compound, start new cycle and go through the diagram again.

Is the residual melt cushion too small (min. 2-5 mm) or does the residual melt cushion fluctuate? *1

No

Do sink marks appear near the gate or in thick-walled areas of the moulded part?

No

Do sink marks appear away from the gate or in thin-walled areas of the moulded part?

No

Do sink marks appear directly after demoulding?

No

1. Increase metering stroke
2. Check non-return valve and/or cylinder
3. Check material feed

1. Optimise holding pressure time
2. Increase holding pressure
3. Change mould wall temperature (-)
4. Change melt temperature (-)
5. Change injection rate (-)

1. Optimise holding pressure time
2. Increase holding pressure (possibly short-term overloading) *2
3. Change injection rate (+)
4. Change melt temperature (+) *3
5. Change mould wall temperature (+) *3

1. Check venting system (especially in the case of high-gloss surfaces)
2. Reduce clamping force *4
3. Check gate and sprue dimensions
4. Introduce flow promoter
5. Check condition of granules (e.g. residual moisture)
6. Eliminate melt accumulations
7. Consider wall thickness/rib ratio
8. Add blowing agent
9. Use plastic with low shrinkage
10. Use plastic with slow cooling characteristics
11. Consider special processes (gas injection moulding, stamping, etc.)

1. Increase cooling time

*1

If necessary, check the residual melt cushion at the aggregate, since some machine controls will not show the smallest screw position, but the screw stroke at the end of the holding pressure time.

*2

Danger of flashing, demoulding problems, mould damages.

*3

Optimise the holding pressure time.

*4

Reduce the clamping force of the machine only to such an extent that – with a safety margin of 20% – there is no risk of overpacking.

CHAPTER 2

Physical causes
and remedies



2.2.1 Charred streaks (brown or silver)

Defect described on page 11



Physical cause

Burnt streaks are caused by thermal damage to the melt. The result can be a decrease in the length of the molecule chain (silvery discoloration) or a change in the macro molecules (brownish discoloration).

Possible causes for thermal damage:

- ▶ Temperature too high or residence time too long during pre-drying
- ▶ Melt temperature too high
- ▶ Shearing in the plasticising unit too high (e.g. screw speed too high)
- ▶ Residence time in the plasticising unit too long
- ▶ Shearing in the mould too high (e.g. injection rate too high)
- ▶ Thermal damage to the colorants

Note:

To check the melt temperature, be sure to shoot "outside" and take the temperature using a needle thermometer. Thermal degradation of the plastic has a negative impact on its mechanical properties, even if no damage is visible on the surface.

2.2.1 Charred streaks (brown or silver)

Defect described on page 11



Investigation

Check and/or change machine settings, change mould or moulding compound, start new cycle and go through the diagram again.

*1
Perform a rheological
mould design.

*2
Thermal damage may
occur due to an exces-
sively long or hot drying
process.

Is the melt temperature above the
processing range?

No

1. Reduce cylinder temperature
2. Reduce screw speed
3. Reduce dynamic pressure

Is the melt residence time within
the critical range?

No

1. Reduce cycle time
2. Increase plasticising time delay
3. Use smaller plasticising units
4. Reduce reclaim part

Are charred streaks appearing
periodically or are they visible
after injecting "to the outside"?

No

1. Avoid dead spots and flow-impeding areas in the gating
system and in the plasticising unit
2. Check plasticising unit for wear
3. Check condition and feed of granules

Do charred streaks appear near
the gate?

No

1. Reduce injection rate (injection profile slow - fast)
2. Check hot runner system
3. Avoid sharp edges in the gating system

1. Reduce injection rate
2. Avoid sharp edges
3. Avoid small flow cross-sections
4. Check sprue and gate system *1
5. Check nozzle cross-section and temperature
6. Check functioning of shut-off nozzle
7. Check pre-drying of material *2
8. Reduce reclaim part
9. Use moulding compound or colouring agents with higher
thermal stability

CHAPTER 2

Physical causes
and remedies

2.2.2 Moisture streaks

Defect described on page 12



Physical cause

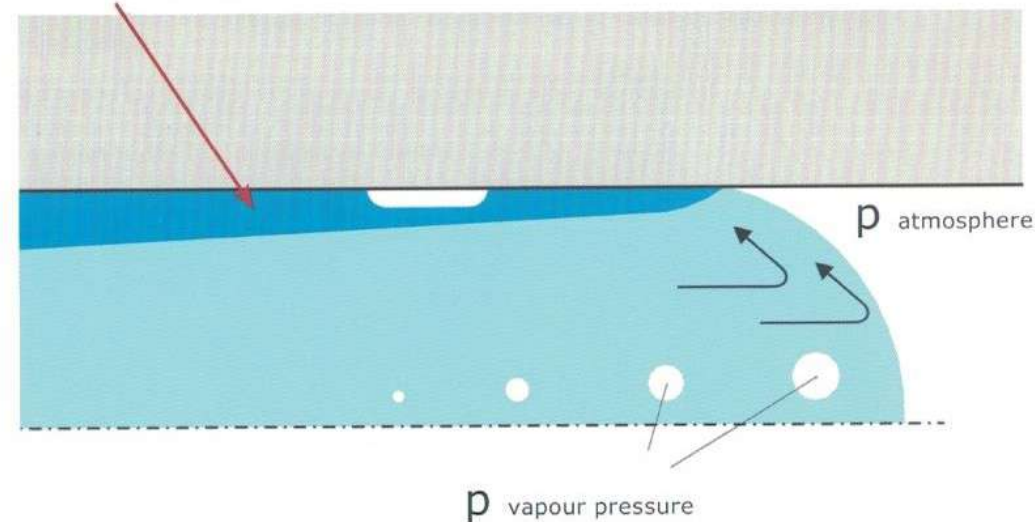
During storage or processing, the granules absorb moisture and form water vapour in the melt. The velocity profile at the flow front transports the gas blisters to the surface of the melt. Due to the effort of compensating for the pressure, the blisters burst, are deformed by the moving flow front and freeze at the mould wall. Possible causes for moisture streaks:

- ▶ Leaky mould temperature control system
- ▶ Condensation water on the mould walls
- ▶ Insufficient pre-drying of the material
- ▶ Inappropriate storage of the material
- ▶ Temperature at flange too low

Moisture on the mould
surface

Moisture in/on the
granules

Solidified peripheral layer



2.2.1

Flow of the water
vapour blisters near
the flow front

2.2.2 Moisture streaks

Defect described on page 12



Investigation

Check and/or change machine settings, change mould or moulding compound, start new cycle and go through the diagram again.

Is there moisture on the mould surface?



1. Check mould cooling system for leaks
2. Increase mould wall temperature
3. Use dry-air-curtain system *1

Reduce moisture content:

1. Sufficiently pre-dry material *2
2. Check material packaging
3. Check storage of material
4. Reduce residence time in material hopper *3
5. Increase temperature at material feed (flange) *4

*1

Application appropriate, if:
► Shorter cooling and cycle times can be achieved by a lower temperature of the temperature control medium
► The optimum temperature of the temperature control medium cannot be maintained due to condensed moisture

*2

Follow the drying instructions provided by the raw material producer.

*3

If possible, feed directly from the dryer in a closed system.

*4

Follow the instructions provided by the raw material producer.

CHAPTER 2

Physical causes
and remedies

2.2.3 Air streaks

Defect described on page 13

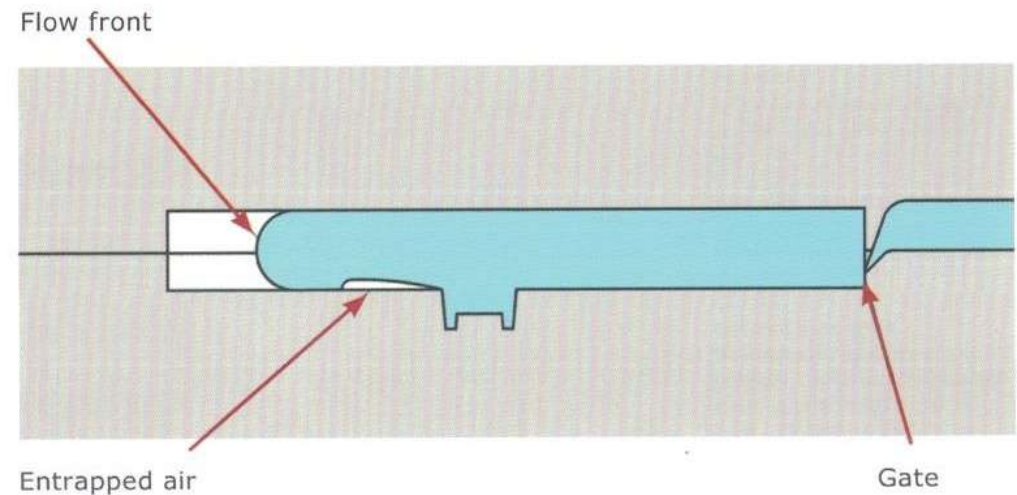


Physical cause

Air which cannot escape in time during the mould filling is drawn to the surface and stretched in the direction of the flow. Especially near lettering, ribs, domes and depressions, the air can be covered and thus entrapped by the melt. The result is the formation of air streaks.

If air is sucked into the area in front of the screw due to decompression (screw retraction), air streaks will usually appear near the gate. Here, the air is transported into the cavity during the injection, and is then pushed towards the mould wall where it freezes.

Air is also drawn in together with the material during the plasticising process. If this air cannot escape in the direction of the hopper, due to an insufficient dynamic pressure or an excessive plasticising capacity, air streaks appear, which are often visible on the entire moulded part.



2.3.1

Formation of an air streak behind an engraving



Investigation

Check and/or change machine settings, change mould or moulding compound, start new cycle and go through the diagram again.

*1
Follow the processing instructions of the raw material and machine manufacturers.

*2
Decrease of the plasticising capacity by means of the screw diameter, the L/D ratio, shear and mixing sections, the use of material-optimised screw geometries (e.g. barrier screws).

Are there air hooks?

No

1. Adapt injection rate (-)
2. Round sharp edges of transitions
3. Reduce depth of engraving

Are there air streaks near the gate?

No

1. Reduce screw return speed during decompression
2. Reduce decompression
3. Use shut-off nozzle

Are blisters visible in the injected melt?

No

1. Increase dynamic pressure *1
2. Check charging and feeding of granules
3. Decrease the plasticising capacity *2

1. Adapt injection rate (-)
2. Provide for venting
3. Round sharp edges of transitions
4. Move gate
5. Check nozzle sealing

CHAPTER 2

Physical causes
and remedies

2.2.4 Coloured streaks

Defect described on page 14



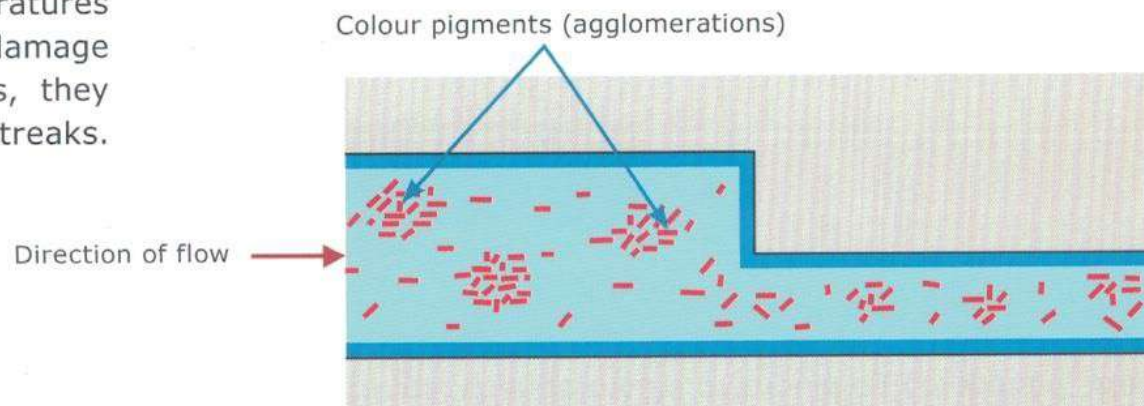
Physical cause

During pigmentation, pigment agglomerations, for example, can lead to differences in the concentration. The plastic, the processing parameters, adhesives and other additives, can cause this kind of poor distribution. With in-plant colouring using dyes, the defect can occur due to incomplete solution of the dye particles in the melt. Similar to thermoplastics, pigments and dyes are sensitive to excessive processing temperatures and residence times. If thermal damage is the reason for colour streaks, they should be handled as burnt streaks.

Extensive strain on the moulded part, e.g. caused by high demoulding forces or warpage, can also cause colour differences. Also in the case of prepigmented materials, homogenizing problems may cause coloured streaks. The deformed areas break the light in a different manner than adjacent areas.

Note:

If using master batches for colouring, make sure the substrate is compatible with the plastic to be coloured.



2.4.1

Decreased differences
in concentration due
to more shearing

2.2.4 Coloured streaks

Defect described on page 14



Investigation

Check and/or change machine settings, change mould or moulding compound, start new cycle and go through the diagram again.

Does the processing range of the material allow higher shearing (danger of thermal damage)?

No

Improve mechanical melt homogeneity:

1. Increase dynamic pressure and speed*1
2. Increase injection rate
3. Use smaller gate and sprue

Are modification of the colouring process possible?

No

Pigmentation:

1. Use smaller pigments
2. Use pigment pastes or master batches *2

Colouring with dye:

1. Use smaller particles
2. Check solubility of the used dye

Can other granules be used?

No

1. Use smaller granules

Changing the machine or the plasticising unit: *3

1. Increase the L/D-ratio
2. Use shearing and mixing sections
3. Use non-return valves with intensive mixers

*1

Follow the processing instructions of the raw material and machine manufacturers.

*2

The MFI-value of the substrate must be identical to that of the plastic.

*3

Ensure sufficient melt homogeneity for the selection of a plasticising unit (shear and mixing sections must not damage plastic and dye).

CHAPTER 2

Physical causes
and remedies

2.2.5 Glass fibre streaks

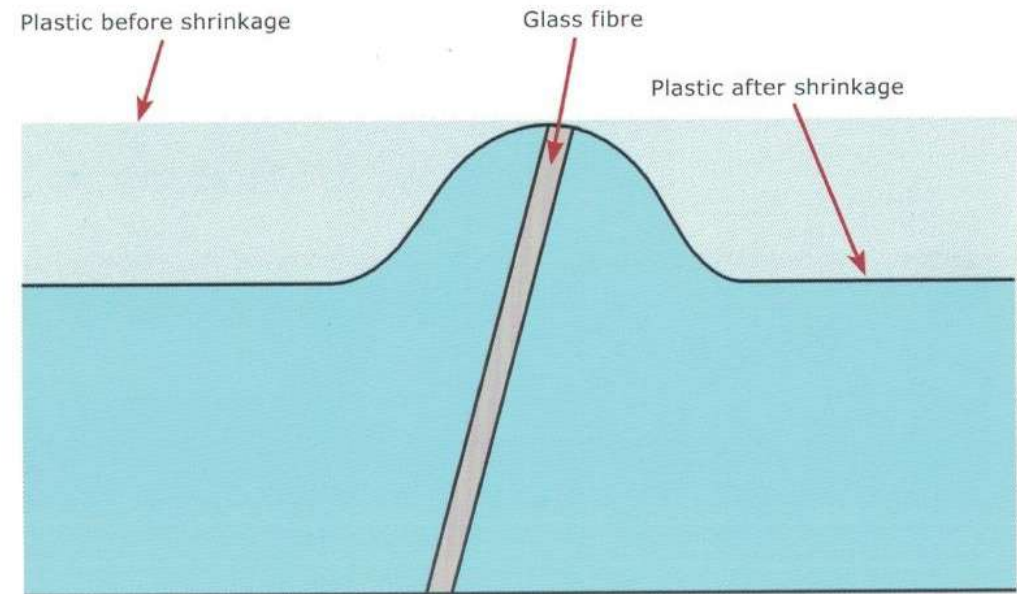
Defect described on page 16



Physical cause

Due to their geometry, glass fibres align in the direction of flow during injection. If the melt suddenly freezes upon touching the mould wall, the glass fibres may not yet be sufficiently surrounded with melt.

In addition, the surface can turn rough because of the large differences in shrinkage (glass fibre : plastic = 1 : 200). The glass fibres impede the shrinkage of the cooling plastic, especially in the longitudinal direction of the fibre, thus producing an uneven surface.



2.5.1

Formation of a rough surface due to shrinkage
differences around the weld line

**Investigation**

Check and/or change machine settings, change mould or moulding compound, start new cycle and go through the diagram again.

Can the process be optimised?

No

1. Increase injection rate
2. Increase mould wall temperature*1
3. Increase melt temperature
4. Optimise holding pressure time
5. Increase holding pressure

Can the material be modified?

No

1. Use shorter glass fibres
2. Use glass beads
3. Use comparable material of the same group

Are there streaks near the weld line?

No

1. Move gate (move weld line to invisible area)
2. Use glass beads

1. Check applicability of two-component process (sandwich moulding process)
2. Use dynamic temperature control *2

*1

Considerable improvement of the defect is possible; considerable negative effect on the cycle time.

*2

See chapter 2.4 "Weld line - dynamic temperature control".

2.3.1 Air entrapments (blistering)

Defect described on page 17

**Physical cause**

During injection, air is entrapped in the melt and appears as a cavity (blister) or in the form of buckling on the moulded part. Primarily, there are three factors responsible for this defect:

- ▶ Decompression excessively high or fast
- ▶ Insufficient dynamic pressure
- ▶ Venting problems inside the mould

In the case of an excessively high or fast decompression, air may be drawn into the space in front of the screw caused by the resulting suction. This air may be entrapped during the injection process. An insufficient plasticising capacity can be attributed to:

- ▶ An excessive metering stroke (should not exceed 3 screw diameters)
- ▶ Difficulties during the material feed

The dynamic pressure builds up a pressure gradient inside the screw during plasticisation. The pressure in the space in front of the screw is very high. It continuously decreases towards the hopper. Due to this pressure gradient, the air drawn in during plasticisation is released again via the feed unit. If the pressure gradient is insufficient, the air is transported into the space in front of the screw during plasticisation and is then injected. In the case of poor mould venting, the air may be entrapped by the melt in areas of, for example, ribs or blind holes.

 Note:

There are two types of cavity- entrapped air and voids. Distinguishing between the two is very hard, because of their similar appearance. Please refer to the defect classification (identification) of blistering and to the chapter on voids.

Moulded parts with cavities are usually not as strong as parts without. Non-transparent parts should therefore be randomly tested by cutting them open.

2.3.1 Air entrapments (blistering) *1

Defect described on page 18



Investigation

Check and/or change machine settings, change mould or moulding compound, start new cycle and go through the diagram again.

Can decompression be reduced?

No

1. Slow down and/or reduce decompression (screw retraction)

Does the defect occur suddenly during processing?

No

1. Check granule charging and feed
2. Check flange temperature *3

1. Increase dynamic pressure and adapt speed*3
2. Use a larger plasticising unit *2
3. Check plasticising unit for function, cleanliness and wear
4. Check temperature profile over cylinder *3
5. Check mould venting

*1
Randomly cut open non-transparent parts, if air entrapments are suspected.

*2
The metering stroke should be 1-3 times the screw diameter.

*3
Follow the processing instructions of the raw material and machine manufacturers.

2.3.2 Voids

Defect described on page 19



Physical cause

Voids appear, if during the cooling of the material the thermal contraction (shrinkage) can no longer be compensated in the corresponding areas. If the outside walls in these areas are already very stable as a result of fast cooling or a favourable geometry, they can no longer be drawn inside by the cooling stresses. The cooling stresses can be high enough to tear apart the incompletely cooled down melt in the inside of the moulded part. The results are evacuated cavities.

Note:

For optimum transfer of the holding pressure, the moulded part must be gated at the largest cross-section. If this is not possible, a flow promoter between the gate and the melt accumulations may improve the holding pressure transfer. In order to avoid premature solidification of the sprue and gate system, sufficient dimensioning is necessary



Investigation

Check and/or change machine settings, change mould or moulding compound, start new cycle and go through the diagram again.

Is the residual melt cushion too small (min. 2-5 mm) or does the residual melt cushion fluctuate? *1

No

1. Increase metering stroke
2. Check non-return valve and/or cylinder
3. Change dynamic pressure (+)
4. Check material feed

Do sink marks appear near the gate or in thick-walled areas of the moulded part?

No

1. Optimise holding pressure time
2. Increase holding pressure
3. Change mould wall temperature (-)
4. Change melt temperature (-)
5. Change injection rate (-)

Do sink marks appear away from the gate or in thin-walled areas of the moulded part?

No

1. Optimise holding pressure time
2. Increase holding pressure
3. Change injection rate (+)
4. Change melt temperature (+)
5. Change mould wall temperature (+)

1. Check sprue and gate dimensioning
2. Introduce flow promoter *2
3. Eliminate melt accumulations
4. Consider wall thickness/rib ratio

*1

If necessary, check the melt cushion at the unit itself. Some machine control systems do not indicate the smallest screw position, but the screw path at the end of holding pressure time.

*2

Introduce a flow promoter between the gate and the melt accumulations for improved holding pressure transfer.

2.3.3 Gaseous entrapments

Defect described on page 20



Physical cause

Gaseous blistering can be attributed to thermal damaging of the material. Thermal damaging of the material due to excessive melt temperatures and/or an excessive residence times, may result in the degradation of the molecule chains or additives.

The decomposition products can be gas-

eous and appear in the form of blisters inside the moulded part or on its surface. This effect can also be caused by extremely long and/or high shearing of the material inside the mould. Thermally damaged material may show subsequent blistering in the case of additional high temperature strains (e.g. soldering).

Note:

With some materials (e.g. LCP, TPE) this defect can also be caused by excessive moisture.



Investigation

Check and/or change machine settings, change mould or moulding compound, start new cycle and go through the diagram again.

Is the melt temperature above the processing range?

No

1. Reduce melt temperature
2. Reduce screw speed
3. Reduce dynamic pressure

Is the residence time within the critical range?

No

1. Reduce cycle time *1
2. Increase plasticising time delay
3. Use smaller plasticising units
4. Reduce reclaim part

1. Check aggregate for wear
2. Reduce injection rate
3. Check sprue and gate systems
4. Check nozzle cross-section and temperature
5. Check pre-drying of material *2
6. Reduce reclaim part
7. Use moulding compound or colouring agents with higher thermal stability

*1

Variations in the residence time due to manual activities; thus periodical occurrence of blisters possible.

*2

Thermal damage may occur due to an excessively long or hot drying process.

CHAPTER 2

Physical causes
and remedies

2.4 Weld line (visible notch, discolouration or gloss difference)

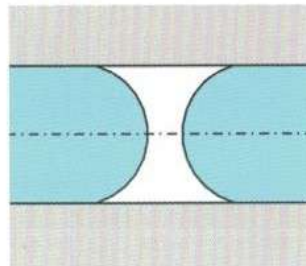
Defect described on page 21



Physical cause

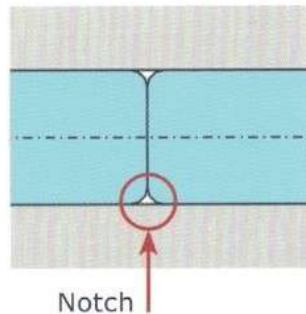
Weld lines are created when two or more melt flows meet. The rounded flow fronts of the melt streams are flattened and bond when touching each other. This process requires stretching of the already highly viscous flow fronts.

If temperature and pressure are insufficient, the corners of the flow fronts will not completely develop. In the case of smooth surfaces a visible weld line notch (see figure 4.2) will occur and in the case of structured surfaces, gloss differences will appear around the weld line. Furthermore, the flow fronts no longer melt together homogeneously, and may produce a weak spot. If moulding compounds containing additives (e.g. colour pigments) are used, strong orientations of these additives around the weld line are possible. This can also lead to colour changes near the weld line.



 4.1

Flow fronts before
touching each other



 4.2

Stretching of the
rounded flow fronts

Note:

Visible weld line notches in the case of smooth or polished surfaces can only be eliminated by high mould wall temperatures. Raising the mould wall temperature, however, increases the cooling time by 20 % per 10° C and can e.g. also cause deviations of the dimensional stability. With the use of dynamic temperature control techniques for eliminating visible weld line notches, it is possible to improve the quality while maintaining economic efficiency in the case of various different applications.



In order to efficiently prevent weld line notches, the mould wall temperature must be increased up to the softening point or the crystallization temperature. The consequence is a temperature increase of the mould wall by 30° C and more, making economic production impossible.

However, new dynamic heating technologies based on water, ceramic heating elements, heating steam, etc., or inductive mould heating offer new methods for reaching mould surface temperatures that are very high either in certain areas or over the entire surface.

Dynamic temperature control enables the prevention of weld line notches of polished or high-gloss polished surfaces under economic points of view.

In the case of polished surfaces, partial heating only in the area of the weld line is often sufficient. Weld lines on structured surfaces require the heating of the entire mould surface since, otherwise, gloss differences would result. The successful use of dynamic temperature control tech-

niques requires extensive mould-, material-, and process-specific knowledge.

In practice, particularly good results have been achieved with amorphous materials such as ABS, PS, PMMA, and PC.

In the case of partially crystalline materials, the concealment of a weld line notch is also possible. However, depending on the type of material, the material might show a rather strong tendency of adhesion to the mould surface. Prior to actually using this technique, material tests should be performed with the aid of a suitable test mould in order to test the suitability of the material in view of the temperature control method. This also applies to other quality features that are to be improved by way of a dynamic temperature control of the mould.

In addition, dynamic temperature control techniques can also be helpful when other attributive surface defects are to be concealed:

- ▶ concealment of partial gloss differences

- ▶ concealment of glass fibre streaks
- ▶ gloss degree adaptation of structured surfaces
- ▶ concealment of dull spots in the gate area
- ▶ concealment of jetting marks on the surface

Note:

The technology described does not drastically enhance the physical properties in the weld line area if the wall thickness is > 1 mm. With a smaller wall thickness, however, strength-increasing effects have been achieved. In addition, it is usually not possible to achieve a different orientation of colour pigments or other fillers, since the orientation of fillers is solely influenced by the flow process and geometry.

CHAPTER 2

Physical causes
and remedies



2.4 Weld line (visible notch, discolouration or gloss difference)

Defect described on page 21



Investigation

Check and/or change machine settings, change mould or moulding compound, start new cycle and go through the diagram again.

*1
High mould wall temperatures can significantly reduce the defect, but drastically increase the cooling time (approx. 20 % / 10 °C).

Are there any notches or gloss differences in the area of the weld line?

No

1. Increase mould wall temperature *1
2. Change injection rate
3. Increase melt temperature
4. Optimise holding pressure time
5. Increase holding pressure
6. Improve venting
7. Move gate (move weld line to invisible area)
8. Use dynamic temperature control (see the preceding description)

Are there discolorations in the weld line area?

No

1. Use smaller pigments
2. Use beads or spherical pigments
3. Use lighter materials
4. Move gate (move weld line to invisible area)

Are there glass fibre streaks in the weld line area?

No

1. Use glass beads
2. Move gate (move weld line to invisible area)
3. Use dynamic temperature control (see the preceding description)

See next page



Investigation

Is the weld line strength insufficient?

No

1. Increase melt temperature and holding pressure
2. Increase mould wall temperature *1
3. Optimise mould venting *2
4. Provide for additional flow through the weld line area (auxiliary cavity)
5. Move the weld line by way of a flow promoter/flow inhibitor
6. Increase the wall thickness *3
7. Use material with a higher weld line stability *4
8. Move the gate

Is there a flow anomaly?

See chapter 26 "Unevenly filled parts – flow anomalies"

*1

Minimal influence of mould wall temperature on the stability. Note increase in cooling time (20 % / 10°C).

*2

Important: sufficient venting around the weld line in the case of soft components.

*3

Increase the cross-sectional area around the weld line.

*4

Often there is still reduced stability around the weld line with the use of plastic material with short fibre strands.

CHAPTER 2

Physical causes
and remedies

2.5 Gloss/gloss differences

Defect described on page 22



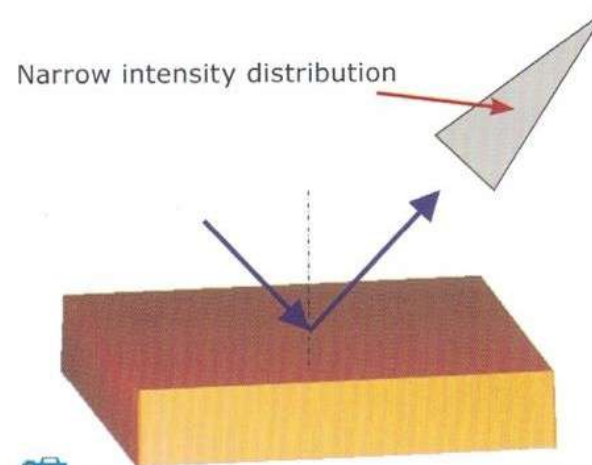
Physical cause

The gloss of a moulded part is the reflecting property of its surface when exposed to incident light. If a ray of light hits the surface of a body, its direction will change (refraction of light). The smoother the surface of a moulded part, the smaller the scattering angle of the reflecting light. The rougher the surface, the larger the scattering angle. The impression of gloss is at a maximum if the surface of the moulded part is as smooth as possible.

To achieve this, a polished mould wall should be projected as well as possible, while a textured mould wall should not be projected as much. Gloss differences are caused by different projection behaviours of the plastic at the mould wall, due to different cooling conditions and shrinkage differences. Stretching

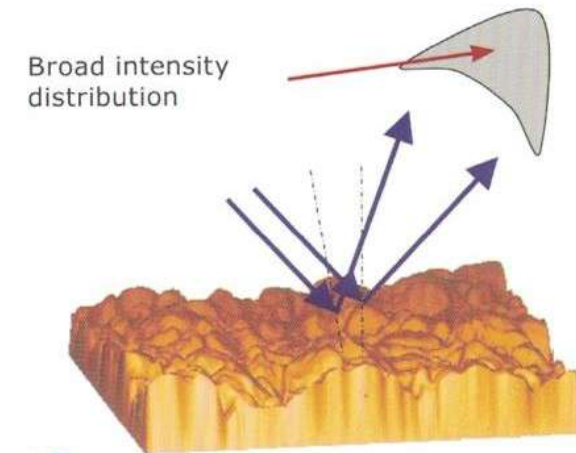
of already strongly cooled down parts of material, e.g. due to warpage, can be another reason for gloss differences. Non-optimal mould venting can also lead to partial gloss differences, particularly

in the case of structured surfaces. They appear in the form of „clouding“.



 5.1

Very glossy impression due to reflection on the polished surface



 5.2

Only slightly glossy impression due to reflection on a rough surface and on filler materials



Note:

Mould coating can affect the degree of gloss on mould surfaces. Due to various influencing factors, a general statement concerning the effects of these coatings cannot be made. Depending on the coating and coating method, the contact temperature and/or mould surface can be affected by a micro-roughness on the original topography.

Dynamic mould temperature control (heating and cooling) can help to conceal the partial gloss differences in the case of structured mould surfaces and, thereby, to produce a uniform degree of gloss, provided that the mould-based requirements are fulfilled. In general, it is also possible to achieve a matt gloss over the entire part surface.



5.3

Change of the degree of gloss due to coating

CHAPTER 2

Physical causes
and remedies



2.5 Gloss/gloss differences

Defect described on page 22



Investigation

Check and/or change machine settings, change mould or moulding compound, start new cycle and go through the diagram again.

Is there insufficient gloss on the polished surface of the moulded part?

No

1. Increase mould wall temperature
2. Change melt temperature (+) *1
3. Increase injection rate
4. Improve polish on mould wall
5. Check the influence of coatings
6. Use dynamic temperature control *2

Are there gloss differences on polished surfaces?

No

Improve thermal melt homogeneity:

1. Reduce residual melt cushion
2. Increase dynamic pressure and adapt screw speed
3. Increase nozzle temperature
4. Check mould surface for deposits
5. Apply even surface polish
6. Check plasticising unit
7. Check uniformity of mould wall temperature
8. Check the influence of coatings
9. Use dynamic temperature control *2

Is there excessive gloss on structured surfaces?

No

1. Increase mould wall temperature
2. Change melt temperature (+) *1
3. Increase injection rate
4. Optimise holding pressure time
5. Increase holding pressure
6. Increase surface roughness
7. Use dynamic temperature control *2

See next page

*1
Reduction only recommended in conjunction with amorphous thermoplastics and high injection rates.

*2
See chapter 2.4 "Weld line – dynamic temperature control".



Investigation

Check and/or change machine settings, change mould or moulding compound, start new cycle and go through the diagram again.

Are there gloss differences at ejectors or slides?

No

Avoid pressure peaks inside the mould

1. Optimise change-over timing
2. Reduce holding pressure
3. Reduce holding pressure time
4. Ensure even mould temperature
5. Change ejector design or system (excessive buckle)
6. Modify slide support

Are there gloss differences at perforations?

No

1. Adapt geometry of perforation (to edge of moulded part)
2. Move gate

Are there gloss differences at the weld line?

No

1. Increase mould wall temperature *1
2. Change injection rate
3. Optimise holding pressure time
4. Increase holding pressure time
5. Improve venting
6. Move gate
7. Increase surface roughness
8. Use dynamic temperature control*2

Are there gloss differences at corners of the moulded part?

No

- Ensure even temperature at corners of the moulded part
1. Reduce temperature of moving half of the mould
 2. Change geometry of corners (e.g. round off corners)
 3. Change thermal mould design

See next page

*1

Raising of the mould wall temperature increases the cooling time by 20 % per 10°C.

*2

See chapter 2.4 "Weld line – dynamic temperature control".

CHAPTER 2

Physical causes
and remedies



2.5 Gloss/gloss differences

Defect described on page 22



Investigation

Check and/or change machine settings, change mould or moulding compound, start new cycle and go through the diagram again.

Is there a gloss differences on
opposite ribs?

No

1. Optimise holding pressure time
2. Increase holding pressure
3. Change the geometry of the moulded part *1
4. Change thermal mould design

Are there gloss differences oppo-
site wall thickness variations?

No

1. Optimise holding pressure time
2. Increase holding pressure
3. Adapt injection profile to geometry
4. Try to obtain continuous wall thickness variations

Gloss difference due to the cloud-
ing of structured surfaces?

No

1. Check and optimise venting system
2. Reduce clamping force of machine *3
3. Reduce injection rate
4. Use dynamic temperature control *2

1. Reduce residual melt cushion
2. Check mould surface for deposits and wear
3. Change colour of material
4. Reduce glass fibre content
5. Reduce filler content

*1
Consider wall thickness/rib
ratio.

*2
See chapter 2.4 "Weld line
– dynamic temperature
control".

*3
Reduce the clamping force
only to such an extent that
– with a safety margin of
20% – there is no risk of
overpacking.



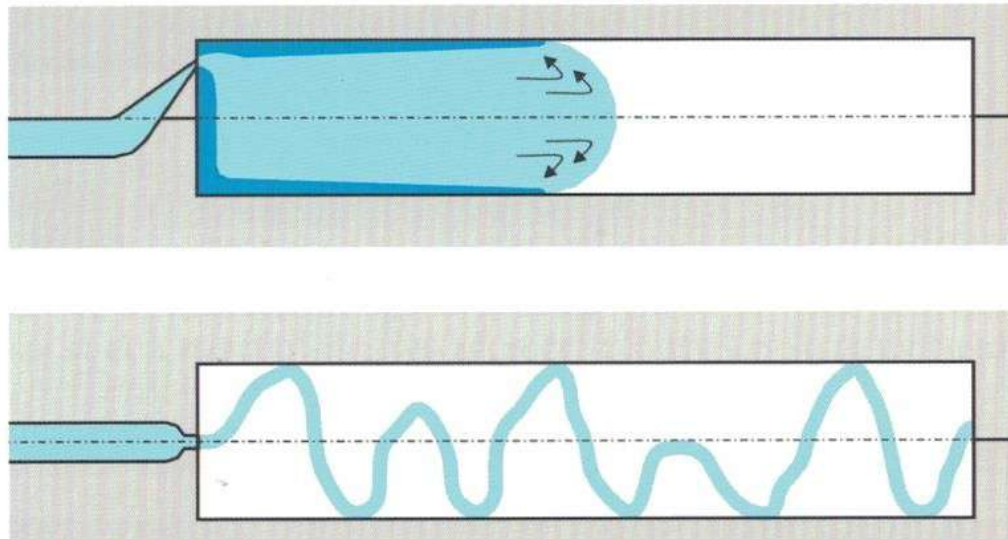
Physical cause

The physical cause for jetting is an undeveloped frontal flow of melt inside the cavity. A melt strand develops, which, starting at the gate, enters the cavity with uncontrolled movements. During this phase the surface of the melt strand

has cooled down to a degree where it cannot bond homogeneously to the rest of the moulding compound. This particularly happens with suddenly increasing cross-sections of the moulded part in conjunction with high injection speeds.

Note:

Jetting can also be influenced by the position of the mould. In order to avoid such defects, the cavity should not be filled from top to bottom.



6.1

Mould filling process
in frontal flow and
with jetting

CHAPTER 2

Physical causes
and remedies



2.6 Jetting

Defect described on page 23



Investigation

Check and/or change machine settings, change mould or moulding compound, start new cycle and go through the diagram again.

Can the injection rate be reduced?

No

1. Reduce injection speed or injection profile (slow-fast) *1

Can the melt temperature be altered?

No

1. Change melt temperature (+)

1. Check position of mould
2. Round off transition gate/moulded part
3. Increase gate
4. Move gate (create flow resistances)
5. Use impact die
6. Use dynamic temperature control *2

*1

In isolated cases the defect can be concealed by means of an extremely high injection rate.

*2

Jetting can be concealed visually by way of dynamic temperature control only by raising the mould surface temperature above the glass transition temperature prior to the injection, e.g. in the case of amorphous materials.



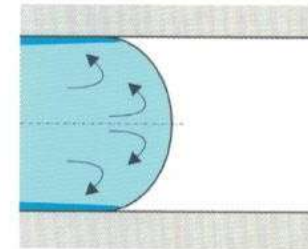
Physical cause

- ▶ Insufficient melt temperature
- ▶ Insufficient injection speed
- ▶ Insufficient mould wall temperature

High cooling rate

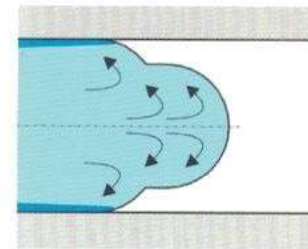
When injecting the moulding compound into a cold mould, a peripheral layer will immediately solidify behind the flow front due to the high cooling rate. The cooling of the peripheral layer also causes flow front areas near the mould wall to cool down. If this happens very fast, especially at low injection rates, these highly viscous or frozen flow front areas can impede the direct frontal flow of the melt towards the wall. Thus the following hot melt will not be transported towards the mould wall as usual, but will cause an elongation of the flow front in the mid-

dle. As of a certain pressure, the flow front will again touch the wall. The heavily cooled down peripheral areas of the flow front no longer completely touch the mould wall.



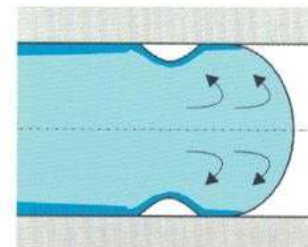
7.1

Flow front has cooled down near the wall



7.2

Cooled down peripheral layer impedes direct frontal flow to the wall



7.3

Flow front touches the mould wall again

CHAPTER 2

Physical causes
and remedies



2.7 Record grooves effect

Defect described on page 24



Investigation

Check and/or change machine settings, change mould or moulding compound, start new cycle and go through the diagram again.

*1
Raising the mould wall temperature increases the cooling time by 20 % per 10° C.

Can the injection rate be increased?

1. Increase injection rate

No

Is there a premature changeover to holding pressure?

1. Optimise change-over point

No

Is the maximum injection pressure reached?

1. Increase maximum injection pressure
2. Use machine with higher injection pressure or higher injection rate

No

1. Increase melt temperature
2. Increase mould wall temperature *1
3. Check sprue system
4. Avoid small flow cross-sections

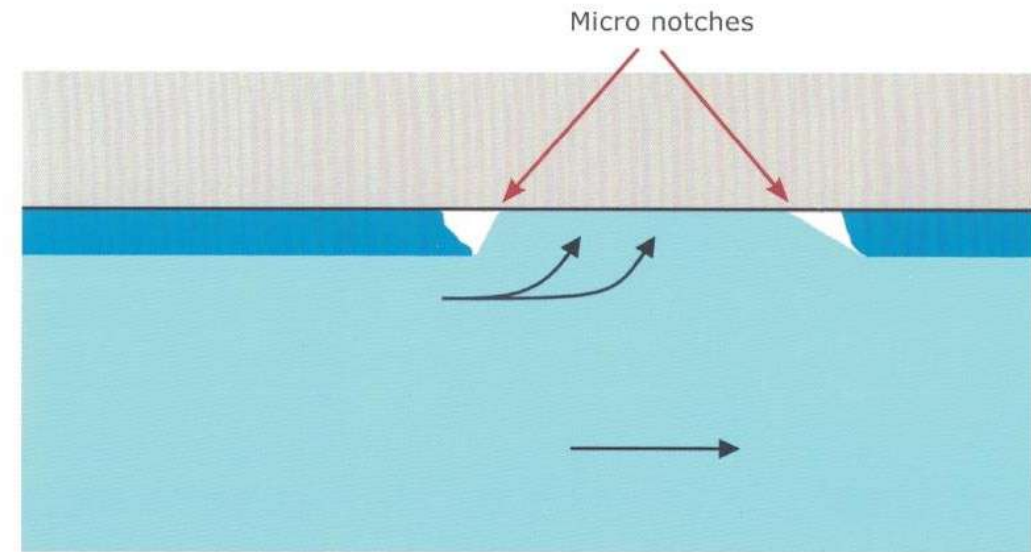


Physical cause

Dull spots in the gate area are primarily caused by:

- ▶ Small gates
- ▶ High injection rates

High injection rates, small gate cross-sections and bypasses behind the gate cause extremely strong orientations of the molecule chains during injection. Since there is not enough relaxation time directly behind the gate, the peripheral layers of the melt are frozen while they are still strongly oriented. Such solidified peripheral layers can only be stressed to a minimal degree and crack under the impact of the high shear stresses. The hot melt inside flows to the mould wall and forms extremely fine notches. The dull appearance is caused by the widely spread reflection in this area.



 8.1

Melt flows into the cracked open peripheral area
(formation of micro notches)

CHAPTER 2

Physical causes
and remedies



2.8 Dull spots near the gate

Defect described on page 25



Investigation

Check and/or change machine settings, change mould or moulding compound, start new cycle and go through the diagram again.

Can the injection rate be reduced?

No

1. Reduce injection rate or injection profile (slow-fast)

Can the gate be modified?

No

1. Round off transition from gate to cavity
2. Increase gate diameter
3. Move gate ^{*1}

1. Increase melt temperature ^{*2}
2. Vary mould wall temperature (-) ^{*2}
3. Use separate temperature control in the area of the dull spot
4. Use dynamic temperature control ^{*3}

^{*1}
Position the gate in such a way that the melt does not have to flow around sharp edges.

^{*2}
The effect of these parameters is small.

^{*3}
See chapter 2.4 "Weld line – dynamic temperature control".



Physical cause

There are several physical causes for incomplete filling:

- ▶ The injected compound volume is too small (e.g. shot volume)
- ▶ The melt flow is impeded due to venting problems
- ▶ The injection pressure or the injection rate of the injection moulding machine is insufficient
- ▶ A channel cross-section freezes prematurely (e.g. insufficient injection rate, wrong temperature control inside the mould or wrong gate position)

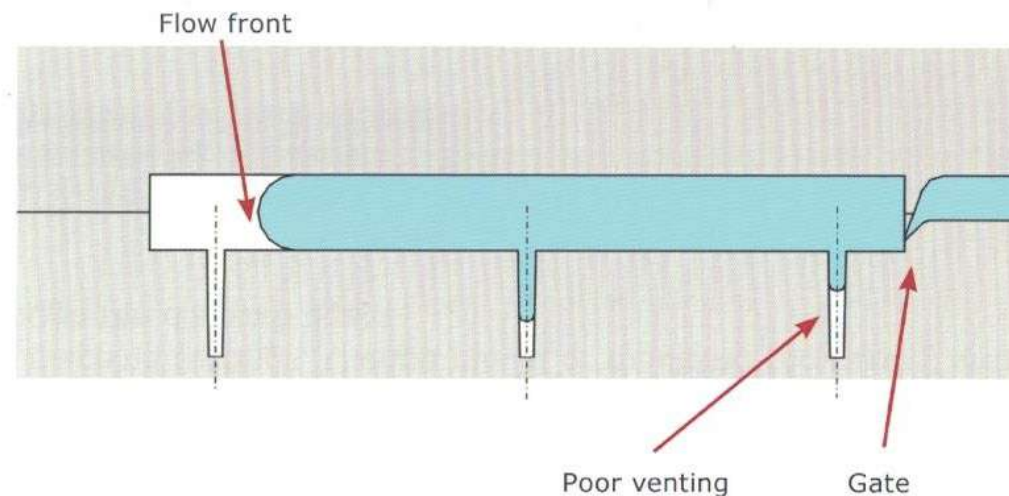
In the case of thin-wall areas near the gate, freezing effects may occur if at the same time the melt can flow into thick-wall areas.

In the thin-wall areas with unfavourable

flow conditions, the melt freezes and thereby prevents a complete filling of the moulded part. Poor venting promotes the occurrence of this defect.

Note:

Incomplete filling due to venting problems will not necessarily cause the diesel effect. Therefore the cause of the defect is often hard to determine.



9.1

Filling problems around thin ribs

CHAPTER 2

Physical causes
and remedies



2.9 Incompletely filled parts

Defect described on page 26



Investigation

Check and/or change machine settings, change mould or moulding compound, start new cycle and go through the diagram again.

*1
Raising the mould wall temperature increases the cooling time by 20 % per 10° C.

Is the screw at the very front?

No

1. Increase dosage
2. Check non-return valve and/or cylinder
3. Check material feed and flange temperature

Is the maximum injection pressure reached?

No

1. Increase maximum injection pressure
2. Increase melt temperature
3. Increase mould wall temperature *1

Is there a pressure drop in the cavity pressure profile (mould filling phase)?

No

1. Optimise changeover point (more delayed changeover to holding pressure)

1. Change injection rate (+)
2. Increase melt temperature
3. Increase mould wall temperature*1
4. Check nozzle aperture and temperature
5. Improve venting
6. Check sprue system
7. Change gate position
8. Use machine with higher injection pressure or higher injection rate



Physical cause

The Diesel effect is a pure venting problem. It can occur near blind holes, fillets, the end of flow paths and near points, where several flow fronts meet. It happens whenever the air cannot escape at all or not fast enough via mould parting surfaces, venting channels or the ejector clearance. Towards the end of the injection process, the air is compressed and thus heated to a high degree. This can result in very high temperatures, which can cause burn marks on the plastic.

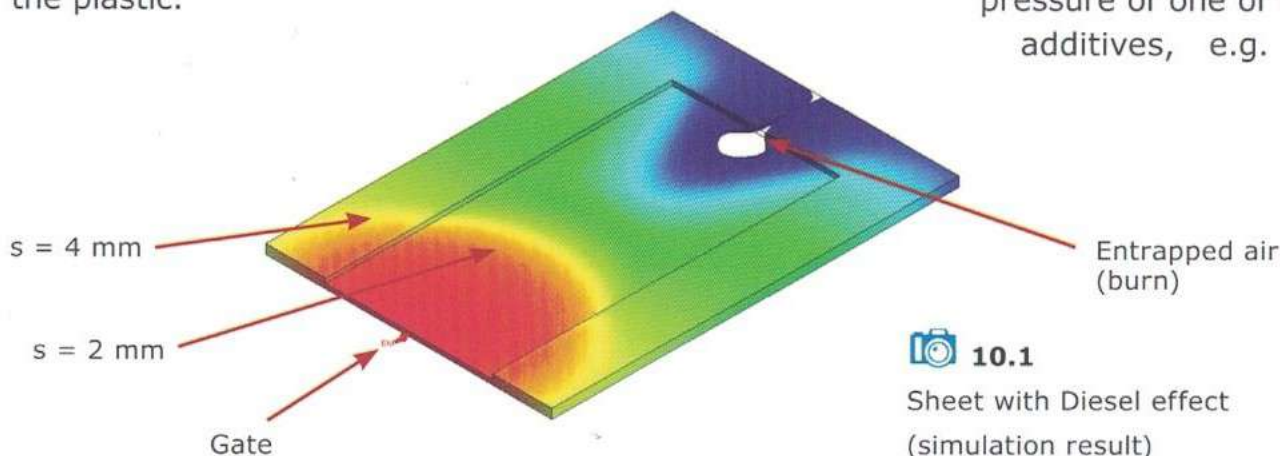
Use of an evacuation system

Conventional venting methods are often not sufficient for the quick venting of a cavity. In this case, an active venting system, a so-called mould evacuation system, can help. In certain cases, the diesel effect can also occur when an evacuation system is used. Due to the vacuum in the cavity and in view of the temperatures that are present during the injection moulding process, the pressure may fall below the vapour pressure of one or several additives, e.g. flame

retardants. This leads to the outgassing of the substances that are compressed by the melt and, thereby, strongly heated. This may cause burns on the part similar to the classic diesel effect.

Note:

The burning of the plastic may result in the formation of aggressive decomposition products, which often damage the mould surface or other mould elements. This can also show in the form of plate-out (see also section 2, page 111) Movable venting elements can be clogged by lubricants (gumming). These problems can be reduced by means of suitable coatings.



CHAPTER 2

Physical causes
and remedies



2.10 Diesel effect/burn

Defect described on page 27



Investigation

Check and/or change machine settings, change mould or moulding compound, start new cycle and go through the diagram again.

*1
Reduce the clamping
force of the machine only
to such an extent that
– with a safety margin of
20% – there is no risk of
overpacking.

Is there no venting in the area of
the burn?

1. Introduce venting possibilities



Does the defect occur suddenly
during production?

1. Check venting channels for contamination



Is the venting installed in the
wrong position?

1. Install venting in right position



Can the clamping force be
reduced?

1. Reduce clamping force of the machine *1



Is there a flow anomaly?

See chapter 26
"Unevenly filled parts – flow anomalies"



1. Ensure sufficient venting
2. Reduce injection rate
3. Avoid entrapped air by changing the flow design
4. Vent mould actively (evacuation)



Physical cause

The possible causes can be classified into four main groups:

- | | |
|--|---|
| ▶ Allowed gap widths exceeded | (Insufficient mould tightness, excessive production tolerances or damaged sealing faces) |
| ▶ Clamping force of the machine is set too high or too low | (Mould opening force higher than clamping force, mould cannot be kept closed or clamping force deforms platens and mould) |
| ▶ Internal mould pressures too high | (Shaping pressure at the gap is so high that the melt is pushed even into very small gaps) |
| ▶ Insufficient viscosity of moulding compound | (High internal mould pressures and low flow resistances promote flash formation) |

Note:

Within a matter of a few cycles, the formation of flashes can damage the sealing faces (mould parting surface), since the local stress, caused by the clamping force, will far exceed the allowed surface pressure of the mould steel.

In order to keep the pressure consumption during mould filling as low as possible, the screw advance speed (injection rate) must be graded (injection profile: slowfast-slow).

Particularly towards the end of the filling, the flow front of the melt will decrease, so that a constant injection rate (constant volumetric flow rate) causes an acceleration of the flow front and thus a pressure increase.

CHAPTER 2

Physical causes
and remedies



2.11 Overpacked parts (burrs, flashes)

Defect described on page 28



Investigation

Check and/or change machine settings, change mould or moulding compound, start new cycle and go through the diagram again.

Are the sealing faces (mould parting surfaces) damaged?

No

1. Repair sealing faces and eliminate cause of damage *1

Can the clamping force be increased?

No

1. Increase clamping force

Is the filling of the moulded part uneven?

No

1. Adapt injection rate
2. Optimise change-over timing
3. Move injection point
4. Adapt the injection times in the case of multiple cavities (e.g. balancing of sprue system)

Are there high mould deformations?

No

Avoid pressure peaks inside the mould: *2
1. Optimise change-over timing
2. Reduce holding pressure
3. Change clamping force
4. Stiffen mould design

Is there overpacking near the sprue?

No

1. Reduce injection rate
2. Injection profile slow-fast

1. Earlier changeover to holding pressure
2. Reduce injection rate or injection profile slow-fast-slow
3. Reduce melt temperature
4. Reduce mould wall temperature

*1
Damages to mould parting surfaces e.g. due to contamination or improper handling.

*2
Ensure proper injection pressure limitation and injection time monitoring particularly in the case of controlled machines.



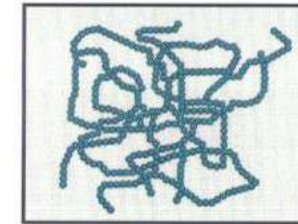
Physical cause

Stress whitening or stress cracks occur as the maximum deformation is exceeded (e.g. due to external stress or warpage). The maximum deformation depends on the type of material used, the molecular structure, the processing and the surrounding climate of the moulded part. The strength against external and internal stresses can be drastically reduced by means of physical processes depending on time and temperature. In this case, wetting, diffusion and swelling processes reduce the bond forces of the molecules. This may especially promote stress cracks. Apart from internal cooling stresses and stresses caused by the flow, internal stresses are often caused by expansion. Internal expansion stress is caused by demoulding under residual pressure, when the moulded part sud-

denly shifts from residual pressure to atmospheric pressure during demoulding: Thus the inner layers of the moulded part put stress on the outer layers. The main reasons for demoulding under residual pressure are insufficiently dimensioned moulds and/or high cavity pressures.

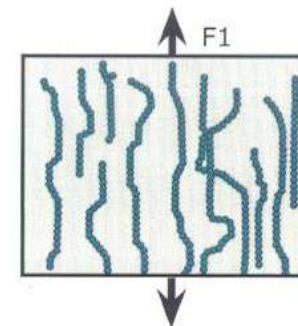
Note:

If aggressive substances are used (e.g. alkali solutions, grease, etc.) stress whitening and stress cracks often appear after



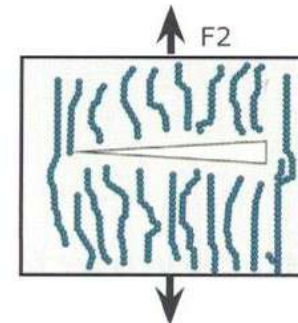
12.1

Unstrained, spatial felt structure



12.2

Molecule orientation due to force F_1



12.3

Torn chains due to additional force F_2

2.12 Stress-whitening/stress cracks

Defect described on page 29



Investigation

Check and/or change machine settings, change mould or moulding compound, start new cycle and go through the diagram again.

Is there stress whitening due to strong deformations?

No

1. Reduce external stress *1

Is there demoulding under residual pressure?

No

1. Earlier change-over to holding pressure (optimise changeover point)
2. Reduce holding pressure
3. Reduce holding pressure time (increase cooling time)
4. Stiffen mould design

Is it a partially crystalline thermoplastic?

No

1. Change mould wall temperature (+)
2. Change melt temperature (+)
3. Reduce holding pressure time or use graded holding pressure profile
4. Reduce holding pressure
5. Change injection rate (+) *2
6. Reduce cooling time *2

Is it an amorphous thermoplastic?

No

1. Select moulding compound under consideration of surrounding medium
2. Use partially crystalline material
3. Try to achieve high molecular weight or narrow molecular weight distribution

Can another moulding compound be used?

1. Ensure even mould temperatures
2. Ensure even filling process
3. Change geometry of moulded part *3

*1
Deformation (e.g. film hinges) in warm condition.

*2
Only for demoulding without residual pressure.

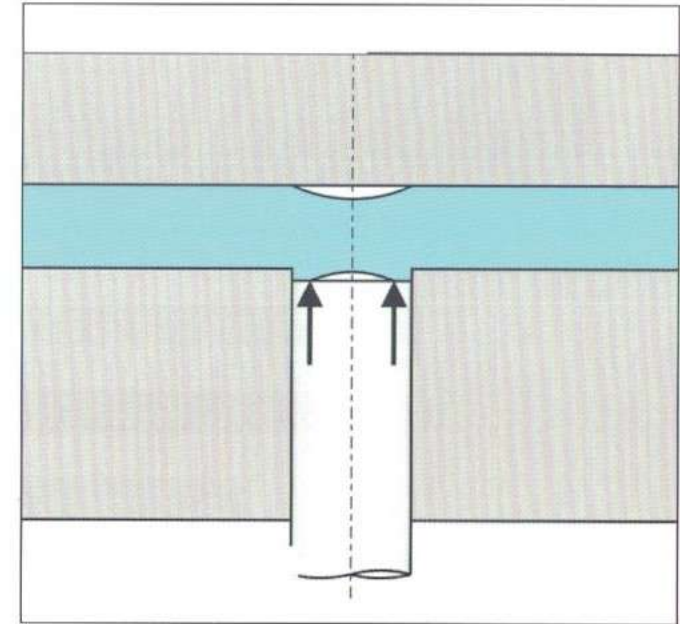
*3
E.g. rounding off edges, avoiding wall thickness variations.



Physical cause

The possible causes can be classified into four main groups:

- ▶ Process-related causes (E.g. premature demoulding or high demoulding forces due to unfavourable machine settings)
- ▶ Geometrical causes (E.g. wrong fitting or ejector length)
- ▶ Mechanical/strength-related causes (E.g. wrong dimensioning and design of the mould, the moulded part or the demoulding system)
- ▶ Thermal causes (High temperature differences within the mould or between the ejector and the mould wall)



 **13.1**

Shrinkage in the area of an overheated and poorly fitted ejector

CHAPTER 2

Physical causes
and remedies

2.13 Visible ejector marks

Defect described on page 30



Investigation

Check and/or change machine settings, change mould or moulding compound, start new cycle and go through the diagram again.

Are there gloss differences?

No

Avoid pressure peaks inside the mould:

1. Optimise change-over timing
2. Reduce holding pressure
3. Reduce holding pressure time
4. Ensure even mould temperature
5. Change ejector design/system (excessive compression of ejector)

Is the ejector not fitted flush or is there an axial allowance?

No

1. Refit ejector
2. Check counterbore and contact surfaces of the ejector heads

Are there projecting ejector marks?

No

1. Use longer ejectors

Is the demoulding too early?

No

1. Increase cooling time

Are the ejector dimensions too small?

No

1. Change ejector design/system

Are there high mould deformations?

No

- Avoid pressure peaks inside the mould:
1. Optimise change-over timing
 2. Reduce holding pressure
 3. Stiffen mould design

Are there high demoulding forces?

No

See section 2.14, "Deformation during demoulding" (high demoulding forces due to shrinkage on the core or heavily ribbed moulded part)

1. Reduce holding pressure
2. Reduce holding pressure time
3. Reduce mould wall temperature



Physical cause

The causes for deformations occurring during demoulding can be classified as follows:

- ▶ The demoulding system cannot apply the forces necessary for demoulding to the moulded part without damaging it
- ▶ The demoulding movement is disturbed (e.g. during tilting inside the mould)

The amount of demoulding force applied is a crucial criterion for both causes and should thus be kept small. Apart from other factors the shrinkage of the moulded part has a direct impact on the demoulding forces. Shrinkage and demoulding forces can be influenced considerably by varying the process parameters. It is to be taken into consideration, however, that the geometry of the moulded part is a very important influencing factor.

In general, low shrinkage is desirable for sleeve and box-shaped parts, since these parts tend to shrink onto the core (= increase holding pressure or reduce cooling time). Near ribs, higher shrinkage reduces the demoulding force, because the ribs are detached from the mould walls (= decrease holding pressure or increase cooling time).

During demoulding under residual pressure the inside of the part is subject to compressive stress, while the surface is subject to tensile stress. This tensile stress can cause stress cracks and deformations. The pressure inside the cavity affects the entire mould. This stress results in deformations at the injection mould. Under the stress, the volume of the cavity increases. If the moulded part solidifying inside the widened cavity does not shrink by at least the amount of this deformation, the deformed mould areas cannot spring back. The item is then jammed inside the mould.

2.14 Deformation during demoulding

Defect described on page 31



A residual pressure builds up inside the moulded part, which is suddenly released during demoulding. It may be impossible to demould the part.

In the case of undercuts, demoulded by means of forced demoulding, the part must be sufficiently elastic to overcome the undercut. In order to avoid any kind of deformation, these elongations must fall below the permissible values. In general, all ductile plastics are better suited for forced demoulding than hard and brittle materials. In some cases, the demoulding can be improved by increasing the ejector rate, in spite of inadvertently large undercuts. The necessary demoulding force is often reduced by a higher surface quality, since micro indentations can be reduced.

However, in the case of certain materials, e.g. PP, PC or TPU, the demoulding force increases in the case of mirror finishes.

Practical experience has shown that some types of plastic strongly adhere to steel surfaces. This often causes significant demoulding problems and interruptions of the production. Materials with an increased tendency to adhere, sometimes in combination with plate-out, are, for example TPEU, PUR and SEBS.

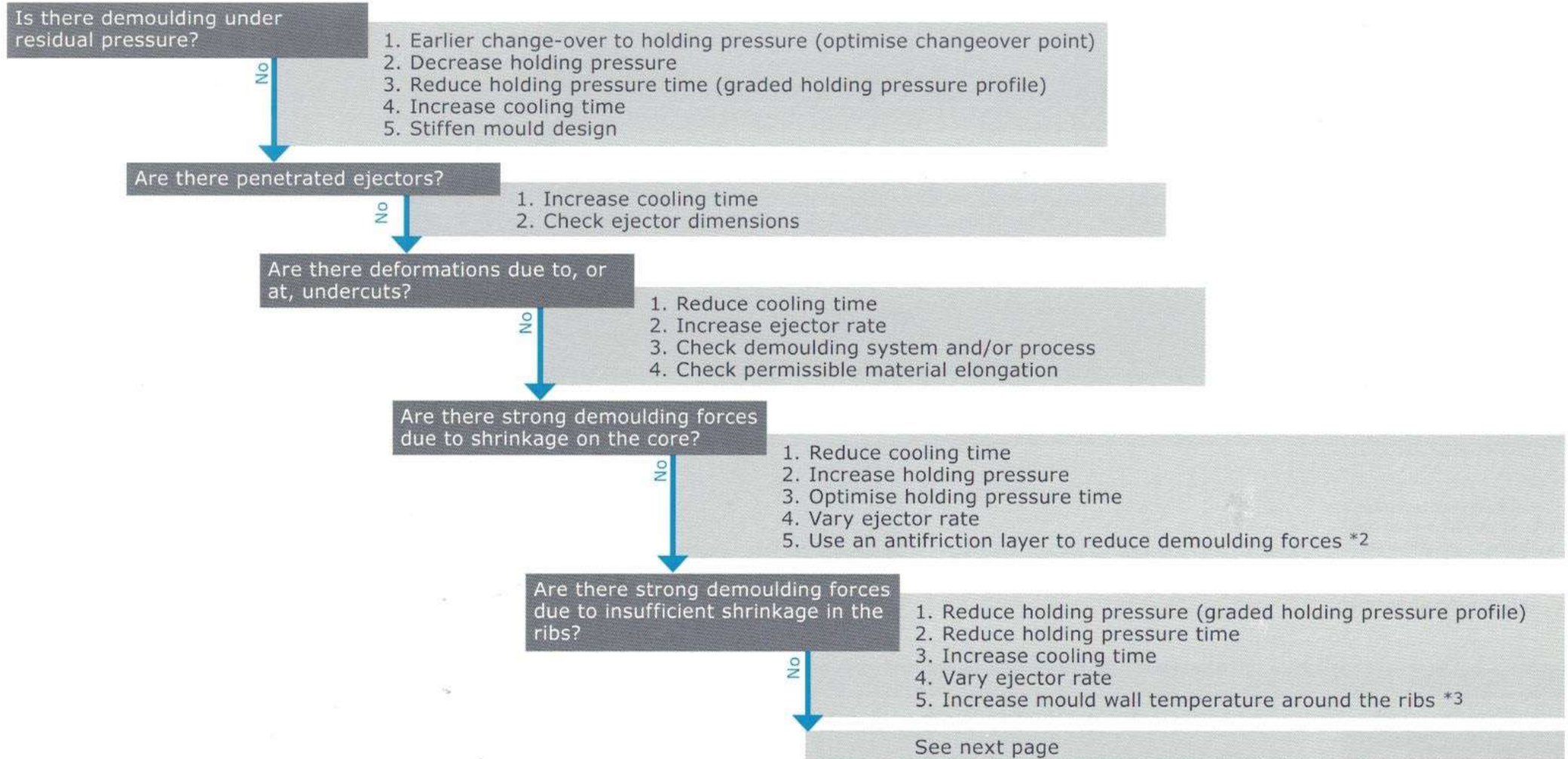
During variothermal injection moulding processes, partially crystalline materials tend to adhere to the mould wall if the mould wall temperature is $> 150^{\circ}\text{C}$.

In such cases, suitable mould coatings can significantly improve the situation.



Investigation

Check and/or change machine settings, change mould or moulding compound, start new cycle and go through the diagram again.



2.14 Deformation during demoulding

Defect described on page 31



Are there strong demoulding forces due to the surface quality?

No

1. Vary mould wall temperature
2. Vary ejector rate
3. Check core venting
4. Check demoulding system
5. Check drafts
6. Use antifriction layers
7. Use mould release agent

*1

Dependence of the surface roughness, regarding the demoulding, on the used moulding compound (see data supplied by raw material manufacturer).

*2

2 Specific layer selection for use of coatings, since some coatings lead to higher demoulding forces.

*3

Raising the mould wall temperature increases the cooling time by 20 % per 10° C.

Are there strong demoulding forces due to adherence of the material to the surface?

No

1. Adapt direction of polish to direction of demoulding
2. Change surface roughness *1
3. Use antifriction layer to reduce demoulding forces *2

Deformation due to adhesion of the plastic material on the surface in the case of dynamic temperature control?

No

1. Reduce mould wall temperature
2. Use coating in order to reduce the adhesion tendency *2
3. Check alternative material

1. Plate-out on the mould surface (see section 20, "Plate-out", page 37/page 111)
2. Check material for additives
3. Air assisted demoulding
4. Change surface roughness *1
5. Use coating to reduce tendency to adhere *2



There is a correlation between the surface structure, the wall thickness and the draft, which influences the demoulding of structured surfaces. In general, the drafts must increase as the surface roughness increases, in order to demould the surface without damage. Also, the drafts can be smaller at identical surface structures but with larger wall thickness due to the higher shrinkage potential. Here, the influence of the shrinkage exceeds that of the drafts. In the case of unfavourable geometric conditions, changes to the processing parameters will not achieve the desired results. If the design allows drawing a core prior to demoulding, smaller wall thickness-draft ratios can be achieved.

The figure below shows the principle correlation between the wall thickness and the shrinkage. Due to the lower wall thickness, ejection grooves may appear

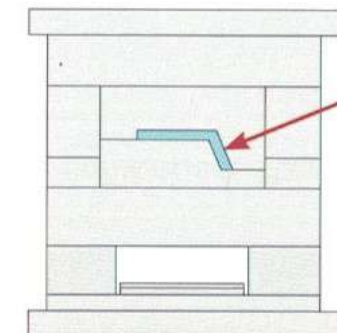
in the lower periphery at identical drafts. In the case of insufficient mould stiffness or excessive mould deformations due to excessive cavity pressures (possibly pressure peaks), ejector grooves may also appear on the lateral surfaces. The reason is the weak dimensioning of the injection mould transverse to the direction of closing. This is then known as an inadvertently large lateral mould cavity expansion. Predestined for this are parts with larger lateral surfaces (see figure 15.2) or sliders

and an insufficient lateral support of the cavity. In many cases the only remedy is a stiffer mould.



 **15.1**

Principle correlation between wall thickness and shrinkage



 **15.2**

Lateral mould cavity expansion

2.15 Ejection grooves

Defect described on page 32



Investigation

Check and/or change machine settings, change mould or moulding compound, start new cycle and go through the diagram again.

Are there ejection grooves on a structured outer surface?



1. Check changeover point *1
2. Reduce holding pressure time (graded holding pressure profile)
3. Reduce holding pressure *2
4. Increase cooling time
5. Increase wall thickness
6. Increase drafts
7. Use lower surface roughness
8. Increase mould stiffness

*1

Pressure peaks inside the mould resulting from a belated changeover to holding pressure, can lead to ejection grooves on the moulded part away from the gate.

This effect may occur if the moulded part is overloaded and the pressure cannot be released in time in the areas away from the gate before the gate sealing point is reached.

*2

The degree of gloss of the surface can change as the holding pressure decreases.



Physical cause

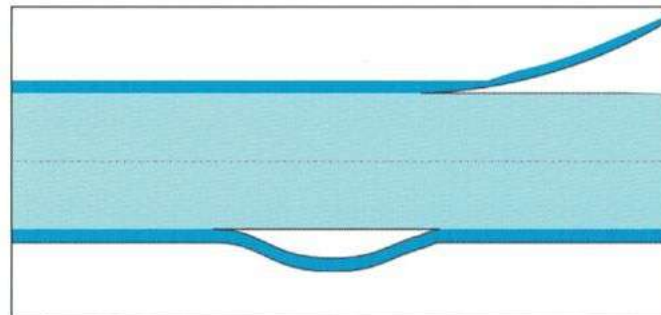
Insufficient bonding of adjacent surface layers causes flaking of these layers. The different layers are formed by different flow effects and cooling conditions over the cross-section. Shear stresses, thermal damage or inhomogeneities can reduce the bonding of these layers to such a degree that single surface layers start flaking off.

Inhomogeneities can be caused by:

- ▶ Impurities or other materials among the granules
- ▶ An incompatible dye or master batch
- ▶ Moisture in/on the granules
- ▶ A poorly melted moulding compound

High shear stresses and thermal damage can be caused by:

- ▶ High injection rates
- ▶ Extremely high or low melt temperatures



16.1

Flaking on the cross-section of a moulded part with different structures

CHAPTER 2

Physical causes
and remedies



2.16 Flaking of the surface layer

Defect described on page 33



Investigation

Check and/or change machine settings, change mould or moulding compound, start new cycle and go through the diagram again.

Does the defect occur after a
change of material or colour?

No



1. Check granules for impurities or other materials
2. Check compatibility of dye
3. Check moisture content
4. Check melt homogeneity and plasticising capacity

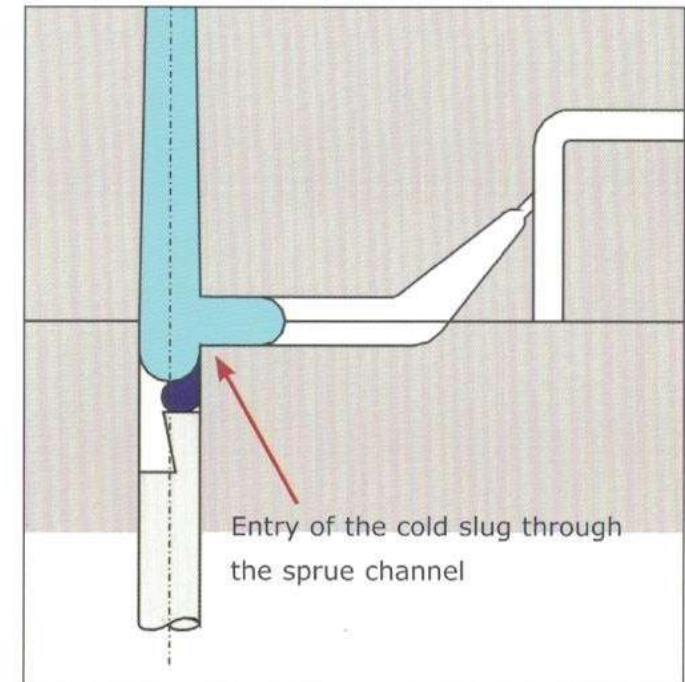
1. Reduce injection rate
2. Vary melt temperature
3. Increase mould wall temperature



Physical cause

Cold slugs are formed when melt solidifies in the gate or in the nozzle, before the mould is filled, and enters the mould with the following shot sequence. If the cold slugs do not melt again, they will cause markings, looking like comet tails. They can be spread all over the moulded part. The cold slug can also jam a runner, forcing the melt to part. The results are surface defects similar to weld lines.

Cold slugs can often be attributed to the wrong nozzle temperature or belated retraction of the plasticising unit. Small nozzle diameters can also have a negative effect.



17.1

The flow front transports the cold slug into the mould

CHAPTER 2

Physical causes
and remedies

2.17 Cold slugs

Defect described on page 34



Investigation

Check and/or change machine settings, change mould or moulding compound, start new cycle and go through the diagram again.

Is decompression used during
production?

1. Optimise decompression



Can the plasticising unit retract
earlier?

1. Retract plasticising unit earlier



1. Check nozzle temperature
(e.g. fastening of strip heaters)
2. Increase nozzle temperature
3. Increase nozzle cross-section
4. Install extended gate extension
5. Use shut-off nozzle



Physical cause

A solidified peripheral layer forms at the moulded part during cooling. This layer is less pronounced in the area of the gate in the case of a hot runner or the transition from the machine nozzle to the gate system. There is a constant heat input, due to which only a very thin peripheral layer can solidify.

A premature movement, e.g. opening of the mould or retraction of the aggregate, can crack this layer open. The material from the area which is still fluid will then be drawn out to form a filament. During the opening movement this filament is extended like a strand, it solidifies and sticks to the moulded part or the gate. The filament can stick to the moulded part and cause a surface defect or it can remain inside the mould and impede the quality of the moulded part during the next shot. Filamentation often occurs with material such as PP or PE or in the case of materials with a narrow processing range (PA, PBT, POM).

CHAPTER 2

Physical causes
and remedies

2.18 Filamentation

Defect described on page 35



Investigation

Check and/or change machine settings, change mould or moulding compound, start new cycle and go through the diagram again.

Is there filamentation in the
area of the gate nozzle?

No

1. Vary nozzle temperature
2. After metering, retract the injection aggregate immediately or with a delay
3. Increase holding pressure time
4. Reduce melt temperature
5. Vary dynamic pressure and screw speed (-)
6. Optimise cooling in the area of the sprue
7. Increase cooling time

Is there filamentation in the
area of the hot runner?

1. Reduce hot runner nozzle temperature
2. Increase holding pressure time
3. Check position of nozzle tip and gate bushing *1
4. Check gate cross-section *2
5. Optimise mould cooling in the gate area
6. Check functioning of temperature sensor and renew, if necessary
7. Check nozzle tip for clogging and replace if necessary
8. Increase cooling time

*1
Large heat input into the moulded part if the nozzle tip penetrates too far into the surface of the moulded part. Gate cross-section too large at retracted nozzle tip.

*2
No clean melt separation, if the cross-section of the gate/ring is too large.



Physical cause

Different factors can cause the formation of dark spots or speckled parts:

- ▶ Process-related causes (E.g. excessive melt temperatures or residence times inside the plasticising unit as well as excessive temperatures in the hot-runner system)
- ▶ Mould-related causes (E.g. dirty gate system or dead spots in the hot-runner system)
- ▶ Machine-related causes (E.g. dirty plasticising system or wear on screw and cylinder)
- ▶ Polymer or dyeing-related causes (E.g. impurities in the granule, high reclaim content or unsuitable dye/master batch)

Note:

(For heat-sensitive materials:)

- ▶ During start-up the allowed residence time of the melt may not be exceeded.
- ▶ In the case of brief interruptions, the temperature decrease is important.
- ▶ In the case of longer interruptions, the plasticising system may have to be cleaned. (Follow the instructions of the material manufacturers).
- ▶ Especially when polycarbonate is used, adhesion and plate-out in the plasticising system may lead to "black spots" on the moulded part. This can be remedied by a surface treatment or a suitable material selection concerning the plasticising system.

2.19 Dark spots

Defect described on page 36



Tips for avoiding "black spots"

- ▶ Prior to the start of the process, flush the system thoroughly with the material that is to be processed. Check for material contamination and continue flushing if necessary.
- ▶ If possible, process interruptions should be avoided or reduced to a minimum by way of suitable measures.
- ▶ In the case of longer production interruptions, reduce the cylinder temperature (follow the instructions of the material manufacturers).
- ▶ At the end of the production run and prior to the cool-down to room temperature, purge the plasticising cylinder sufficiently with a suitable cleaning granulate or purging agent (e.g. PP).
- ▶ In the case of particularly long production times, clean the cylinder, screw, and non-return valve mechanically at regular intervals.



19.1

Black spots after
cleaning of the
plasticising unit



Investigation

Check and/or change machine settings, change mould or moulding compound, start new cycle and go through the diagram again.

Are there impurities in the granules?

No

1. Prevent impurities in the granules
2. Clean plasticising unit
3. Avoid using reclaim
4. Use material deduster

Do dark spots appear after a change of material?

No

1. Clean plasticising unit *1

Does the melt temperature exceed the processing range?

No

- Reduce melt temperature:
1. Vary cylinder temperature (-)
 2. Vary screw speed (-)
 3. Reduce dynamic pressure

Is the melt residence time within the critical range?

No

1. Reduce cycle time
2. Increase plasticising time delay
3. Reduce cycle time
4. Use smaller plasticising units

1. Check hot-runner temperature
2. Reduce reclaim part
3. Check dye for compatibility
4. Check plasticising system, gate system and hot-runner for impurities, wear and dead spots
5. Check material selection and surface treatment of plasticising system

*1

Cleaning of the plasticising unit by means of low-pressure blasting and suitable blasting media (no sand) or by brushing (no steel). Avoid scarfing, since an oxide layer may develop, which will cause the plastic to adhere even more during application and since the screw may be damaged as a result of thermal stress (stress, warpage or differences in structure and hardness).

2.20 Plate-out

Defect described on page 37

**Physical cause**

Plate-out is a common symptom with the processing of thermoplastic materials. It more frequently occurs in materials such as POM, PP, PET, ABS, PC, PSU, PBT and PE. Materials with additives such as UV absorbers, flame retardants, lubricants and colorants are also subject to plate-out. Precipitation of colorants can usually be attributed to the incompatibility of the combination of lubricants and colorants. The colorant is transported on the lubricant as a kind of "slide bar" and settles on the mould surface. This process is called "plate-out". In the case of white pigments the precipitation is called "chalking" and can be attributed to an oxidative decomposition of the plastic. Materials containing flame retardants may chemically react if the stress on the material gets too high. The resulting decomposi-

tion products can cause deposits on the mould and, depending on the design, also corrosion inside the mould.

Plate-out is caused by:

- ▶ High material stresses, such as excessive melt temperatures, shear stress and shear rates
 - ▶ A large amount of shearing inside the plasticising unit, e.g. due to excessive screw speeds
 - ▶ Excessive residence times
 - ▶ A large amount of shearing inside the mould, e.g. due to excessive injection rates
 - ▶ Insufficient mould venting (perhaps due to excessively high clamping forces)
- ▶ Use of materials with excessive residual moisture
 - ▶ Wrong or excessive use of lubricants
 - ▶ Incompatibility of base material and colorant/additive
 - ▶ Excessive drying times



It is often impossible to eliminate plate-out by single optimisation measures. In particular in the case of processes bordering on certain limits, a combination of numerous optimisation steps that are described in this guide is promising in preventing plate-out in the mould and in minimising the cleaning intervals.

Suitable strategies for the process start-up and the conduct in the event of process interruptions are helpful in preventing plate-out due to the thermal degradation of the melt in the hot runner and in the plasticising unit.

When working with certain POM types, plate-out can be reduced and the surface quality can be increased by predrying at approximately 80°C for two hours.

Cleaning of injection moulds

Due to the ever-increasing requirements that are placed on plastic materials, which in turn require the use of additives, plate-out cannot be completely prevented. An effective way of cleaning injection moulds in a machine is the use of dry ice.

Note:

To check the melt temperature, make sure to shoot "outside" and take the temperature using a needle thermometer. If lubricants are used, the thermal and chemical properties must be taken into consideration.

General notes

1. Vary the injection rate (–)
2. Avoid sharp edges and small flow cross-sections
3. Do not exceed the allowed drying times
4. Avoid entrapped air by means of a suitable flow design
5. Check the sprue and gate system
6. Check the nozzle cross-section
7. Check the hot runner system
8. Check whether surface and layer technologies can be applied

CHAPTER 2

Physical causes
and remedies



2.20 Plate-out

Defect described on page 37



Investigation

Check and/or change machine settings, change mould or moulding compound, start new cycle and go through the diagram again.

Does the melt temperature exceed the permissible values?

No

Reduce melt temperature:
1. Vary cylinder temperature (-)
2. Vary screw speed (-)
3. Reduce dynamic pressure

Is the residence time of the melt too long?

No

1. Reduce cycle time
2. Increase plasticising time delay
3. Use smaller plasticising units

Is there too much moisture in the material?

No

Reduce moisture content:
1. Pre-dry material following manufacturer's instructions
2. Check packaging and storage
3. Reduce residence time in material hopper
4. Increase temperature at material feed (flange)

Is the venting of the mould insufficient?

No

Ensure sufficient venting:
1. Check venting channels for contamination
2. If possible, reduce the clamping force
3. Reduce injection rate
4. Rework venting (position and size)
5. Evacuate the mould

Has the wrong or excessive lubricant been used?

No

1. Use suitable lubricant
2. Check metering of lubricant
3. If necessary, use antifriction layers for moving mould elements

Are colorants or additives blended in?

No

1. Check compatibility of base material/colorant/additive
2. Check thermal stability of additives
3. Check mixing ratio

See preceding page (general notes)



Film hinges are moving and permanent joints. Their function is based on the elastic properties of the material. Film hinges are designed according to the functional demands and the following criteria:

- ▶ Bending angle
- ▶ Number of load cycles
- ▶ Load frequency
- ▶ Forces to withstand

The highest mechanical load is applied in the case of large bending angles (180°) and high numbers of load cycles. Apart from the material characteristics, the proper functioning of a film hinge also largely depends on the design and the processing parameters of the injection moulding machine. The position of the

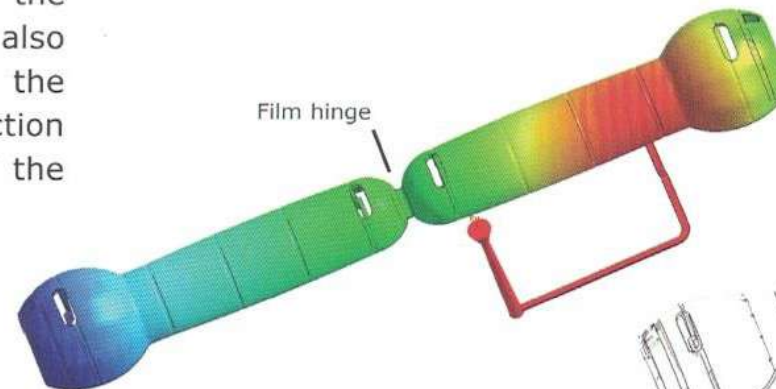
gate is very important. The gating position must ensure an unimpeded flow through the film hinge.

The selection of the gating position is important in preventing the development of weld lines, short-term melt flow standstill and partial freezing effects at the film hinge.

If possible, the flow front should flow in parallel through the film hinge (see figure).

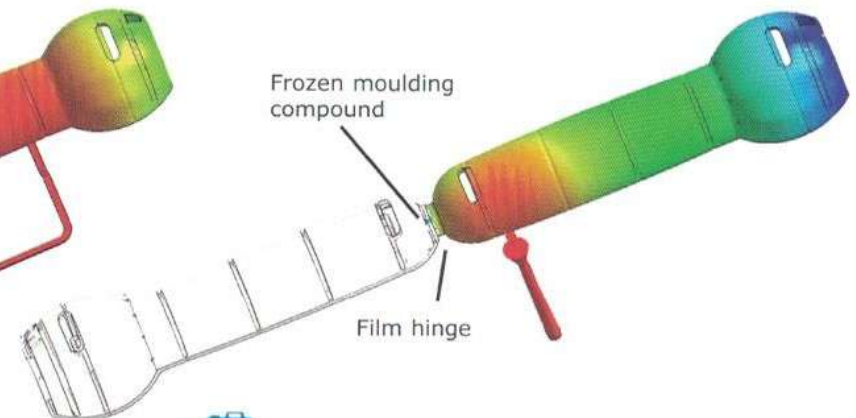
Note:

In order to prevent melt flow standstill or freezing effects in the film hinge, the gate should be positioned so that the large volume part of the moulded part will be filled before the flow front reaches the film hinge. A reliable statement regarding the most suitable gating position can only be made with the help of a simulation.



 **21.1**

Gating away from the film (completely filled)



 **21.2**

Gating close to the film (not completely filled)

2.21 Broken/ragged film hinge

Defect described on page 38



Investigation

Check and/or change machine settings, change mould or moulding compound, start new cycle and go through the diagram again.

Is the film hinge not filled?

No

1. Increase injection rate
2. Increase melt temperature
3. Increase mould wall temperature
4. Increase wall thickness of film hinge *1
5. Use easier flowing material
6. Move gating position away from film hinge *2

Is too much force required to move the film hinge?

No

1. Reduce wall thickness of film hinge
2. Use material with a lower modulus of elasticity
3. Check film hinge design

Does the film hinge break immediately or after just a few uses?

No

1. Remove weld line from film hinge area
2. Increase injection rate
3. Change mould wall temperature (-) *3
4. Overstretch film hinge directly after the injection-moulding
5. Use material with higher viscosity
6. Ensure parallel flow of flow front through
7. Check film hinge design *4

Does stress-whitening develop on the film hinge?

1. Actuate film hinge in warm condition

*1
Consider maximum material elongation.

*2
Melt flow standstill or freezing effects due to gating positions in the vicinity of the film hinge.

*3
More elastic material behaviour of partially crystalline materials in film hinges possible by reducing the mould wall temperature.

*4
Function impairment due to excessive material elongation and film hinge design.



Warpage on injection-moulded parts is a large problem, as it is impossible to precisely predict it. Modifications to injection moulds often cause unpredictable costs and delays. Warpage is always the result of different shrinkage values in a moulded part.

Shrinkage

Shrinkage is a decrease in the volume of the moulded part during the cooling phase. If a plastic melt (figure 22.1a) cools down, its volume will decrease due to the molecule chains nearing each other (22.1b). In the case of partially crystalline materials, the shrinkage is higher compared with amorphous materials as a result of the crystallisation process, during which the molecule chains assume a very high packing density (22.1c).

Shrinkage is classified into:

- ▶ Moulding shrinkage (dimensional changes after 16 hours, see DIN 16742)
- ▶ Aftershrinkage (dimensional changes between the end of the moulding shrinkage and after a long time)
- ▶ A-Total shrinkage (moulding shrinkage and aftershrinkage)

Structure of the plastic materials:



22.1a

Melted phase



22.1b

Solid phase
amorphous

Partially crystalline areas

Amorphous areas



22.1c

Solid phase
partially crystalline

CHAPTER 2

Physical causes
and remedies

2.22 Warpage

Defect described on page 39

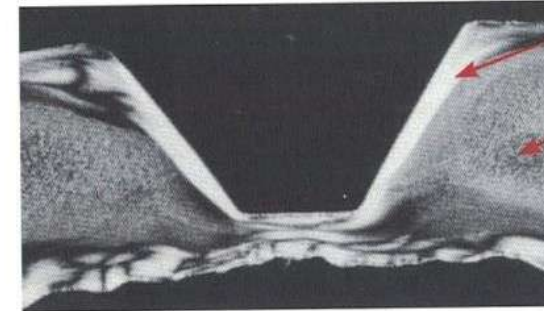


Major influences on shrinkage

1. Mould wall temperature

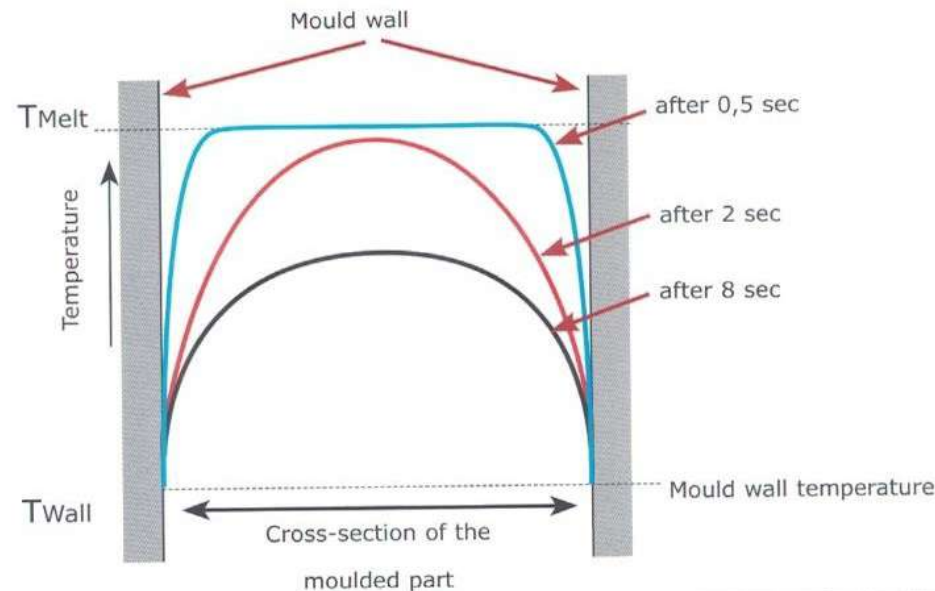
Fast cooling rates in areas near the wall (figure 22.2) do not leave sufficient time for the molecules to assume a high packing density. In addition, in the case of partially crystalline thermoplastic materials, the formation of a crystalline structure in the peripheral areas is suppressed. It can almost remain amorphous there, as shown in figure 22.2. There is little shrinkage in these areas near the wall. In contrast to these peripheral areas, however, the cooling rate in the middle of the cross-section of a moulded part is much lower (figure 22.3), since these areas are insulated by the areas near the wall. In the case of partially crystalline thermoplastics a crystalline structure can develop (see figure

22.2), which leads to higher shrinkage values than in the areas near the wall.



"Amorphous"
peripheral layer

"Partially crystalline"
structure



22.2

Structure of a partially crystalline thermoplastic material)

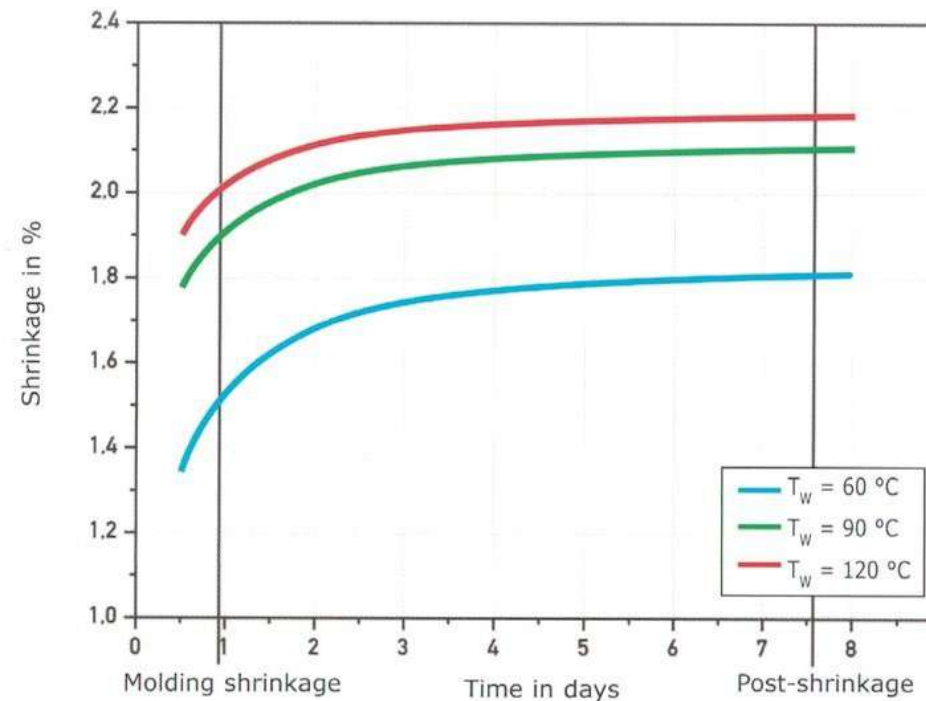
22.3

Temperature profile over the cross-section of the moulded part (example)



The mould wall temperature also has a large impact on the shrinkage, as shown in figure 22.4.

- ▶ A low mould wall temperature causes fast cooling of the moulded part, thereby suppressing the formation of a high packing density of the molecule chains. The moulded parts show little moulding shrinkage, but a lot of after-shrinkage. This particularly applies to partially crystalline materials.
- ▶ A high mould wall temperature causes slow cooling of the moulded part. The molecule chains have sufficient time to assume a high packing density. The moulded parts show little moulding shrinkage, but a lot of aftershrinkage. The total shrinkage is always higher in the case of high mould wall temperatures compared with low mould wall temperatures, if the demoulding temperature is kept constant.



22.4

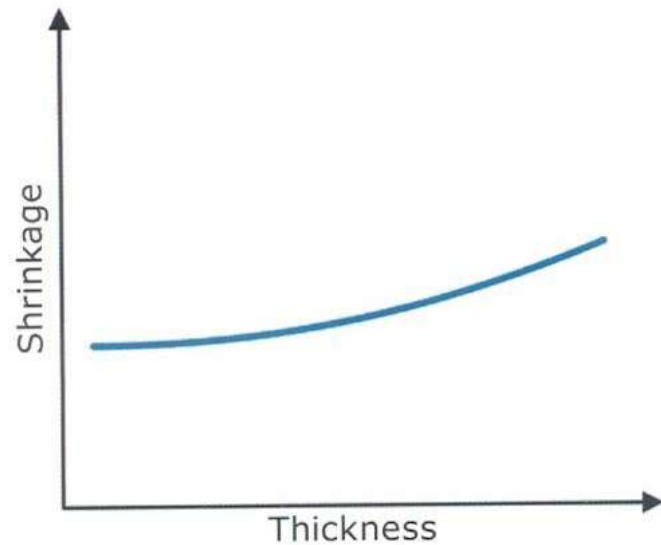
Influence of the mould wall temperature on the shrinkage of a PA6

2.22 Warpage

Defect described on page 39

**2. Wall thickness**

Areas with a large wall thickness cool down slower than those with thin walls, thereby increasing the shrinkage (figure 22.5).

**22.5**

Shrinkage values
depending on the
wall thickness (sche-
matic)



3. Material

There is a difference between amorphous plastics (e.g. ABS, PC, PMMA, PS, SAN) and partially crystalline plastics (e.g. PP, POM, PA, PE).

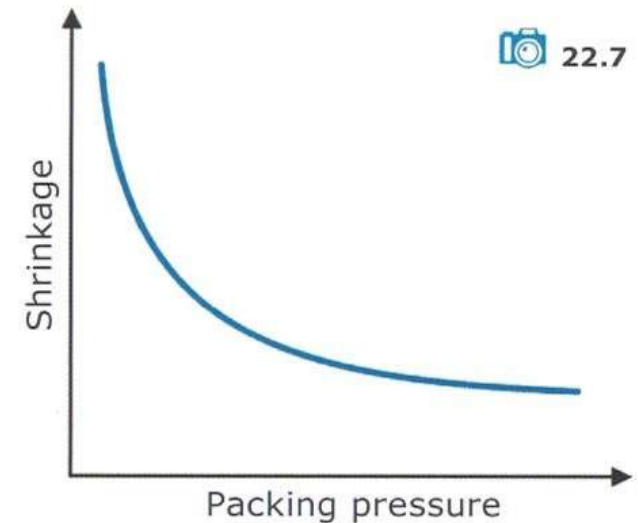
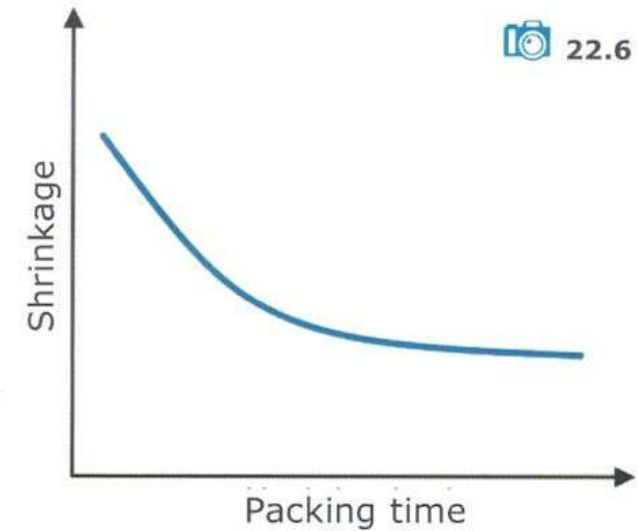
Amorphous plastics usually have a lower shrinkage potential than partially crystalline materials. In their direction of orientation, extenders and reinforcing fillers can impede the shrinkage caused by flow processes inside the mould.

The shrinkage ratio between a partially crystalline plastic and a glass fibre is approx. 200:1. The use of glass fibre can therefore cause enormous shrinkage differences in a moulded part, depending on the position and number of gates, the wall thickness, the geometry of the moulded part, etc. Plastics filled with minerals or glass beads lead to less shrinkage differences due to their geometry. They are often used in combination with glass fibres, to reduce shrinkage differences and thus warpage.

4. Processing parameters

The process also influences the shrinkage behaviour (depending on the plastic used) more or less intensely.

The following section qualitatively describes some dependencies of the major processing parameters and their influence on the shrinkage.



CHAPTER 2

Physical causes
and remedies

2.22 Warpage

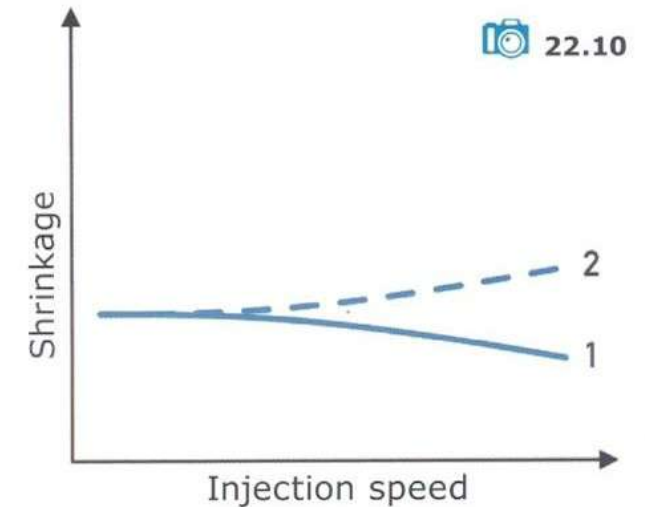
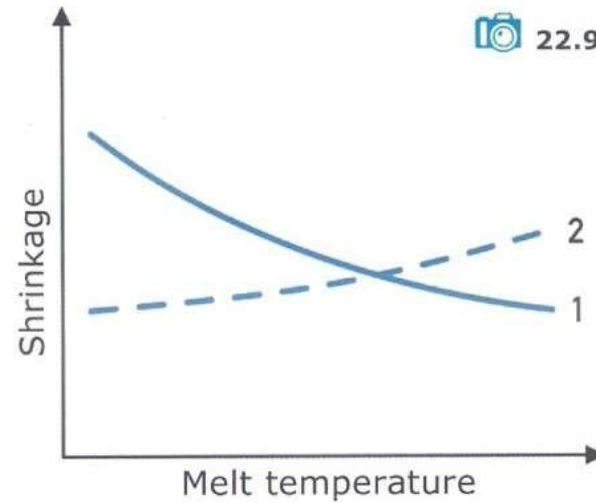
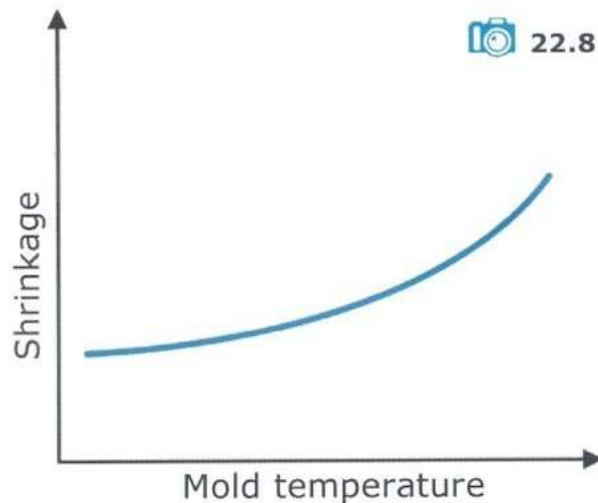
Defect described on page 39



The melt temperature and the injection rate can have a different impact on the shrinkage, depending on the corresponding moulded part, since increasing these parameters improves the flow and thus the pressure transfer, but at the same time the shrinkage potential increases as a result of the higher temperature.

Orientation aid:

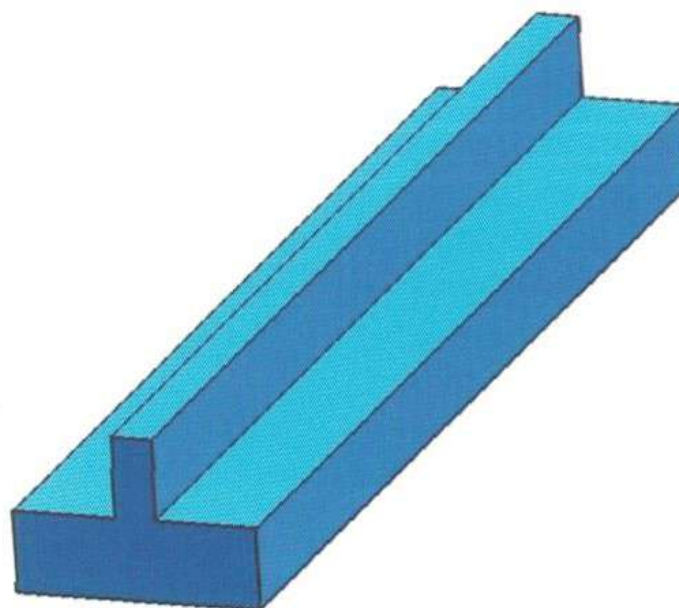
- ▶ For moulded parts with a large flow path/wall thickness ratio (usually thin-wall parts) these parameters should be set high (1).
- ▶ For moulded parts with a small flow path/wall thickness ratio (usually thick-wall parts) these parameters should be set low, to reduce shrinkage and thus warpage (2).





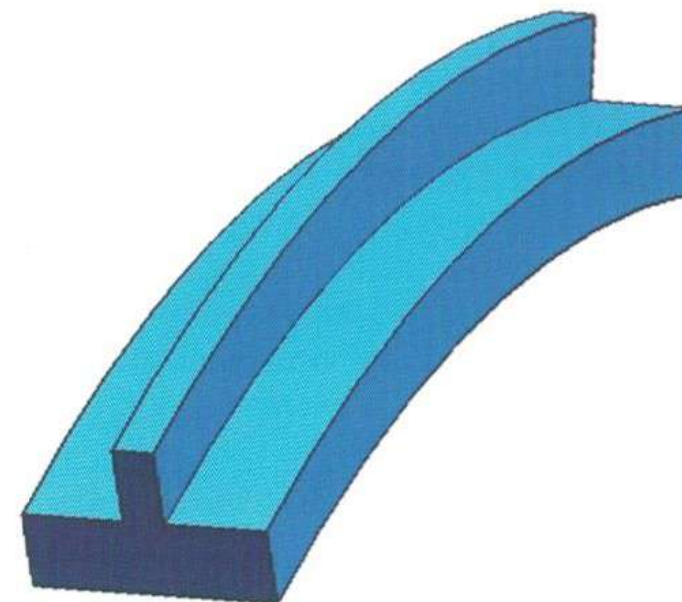
Correlation between shrinkage and warpage

In general, warpage is the result of different shrinkage values within a moulded part. If the thick wall in the figure above shrinks more than the thin rib, the moulded part will warp as shown in the picture on the right.



 22.11

Cavity



 22.12

Moulded part after
demoulding

2.22 Warpage

Defect described on page 39

**Remedies:**

Only qualitative hints to provide warpage can be supplied here.

Design aspects

- ▶ Provide for an even wall thickness
- ▶ Use walls and ribs of the same thickness (possibility of sink marks, gloss differences, etc.)
- ▶ Avoid melt accumulations and sudden changes in the wall thickness
- ▶ Round off corners and increase the size of inner surfaces, to achieve even heat dissipation
- ▶ If possible, base and supporting surfaces should not be flat but dished or profiled
- ▶ Prefer symmetrical designs
- ▶ If possible, move the gate to the thickest area (wedge principle)
- ▶ Choose a gate diameter of a sufficient size (transfer of holding pressure)
- ▶ Provide for an even direction of flow (orientation of glass fibres) inside the moulded part by means of the number of gates and their positions and avoid weld lines

Selection of the moulding compound

- ▶ Use low-warpage plastic materials
- ▶ Consider the influence of the fillers depending on the type and the amount used

Mould design

- ▶ Ensure a homogenous mould wall temperature control
- ▶ Ensure a good thermal separation of hot runner nozzle and moulded part
- ▶ Install separate temperature control circuits in the mould
- ▶ Check the use of thermally conductive materials, if conventional temperature control is impossible



Physical cause

In multi-component technology, at least two different melts are fed into the mould during the injection moulding process. This can be done using the same material but in different colours, or using different materials.

With the exception of in-mould assembly, the melts should have sufficient adherence to each other. Adherence is achieved by fusing one component with the next, and can generally be explained by the diffusion theory. There also has to be mechanical as well as thermal compatibility regarding existing shrinkage differences and the coefficient of thermal expansion so as to make sure the adhesion will withstand thermomechanical stress. Due to the multitude of factors influencing the adhesion (e.g. geometric design of the contact surfaces, processing parameters and the number of material modifications within a material group) the matrix shown on page 125 must be examined closely. In any case, the processor should get in touch with the raw material manufacturer.

In the following multi-component technology will be explained using 2-component technology as an example. The first component corresponds to the substrate and the second to the finished product.

The corresponding mould technologies are classified into turning and non-turning mould systems:

- ▶ Rotating mould
- ▶ Index plate (turnstile)
- ▶ Stack turning technique
- ▶ Transfer technique
- ▶ Core-back technique

Note:

For the elimination of defects, the first step should be to look at each component individually and to apply a remedy suggested in this guide.

CHAPTER 2

Physical causes
and remedies

2.23 Multi-component technology

Defect described on page 40



		Overmould / Finished part																																			
		Thermoplastics																			Flexible components																
		ABS	ASA	CA	EVA	PA 6	PA 6.6	PA 6.6	PA 12	PBT	PC	PC/ABS	PC/PBT	PC/PET	PE-HD	PE-LD	PETP	PMMA	POM	PP	PPE	PPSU	PS	PSU	PVAC	PVC-hard	PVC-soft	SAN	TPE-A	TPE-E	TPE-O	TPE-S	TPE-R	TPE-U	TPE-V	TPE HM	
Pre-mould	Thermoplastics	ABS	+	+	+		+	+				+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	
		ASA	+	+	+	+		+			+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	
		CA	+	+	+	+					+					+	+			+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+
		EVA		+	+	+										+	+				+		+				+										
		PA 6	+				+	+			+	+	+	+	+	+	+			+	+	+	+	+					+	+	+	+	+	+	+	+	
		PA 6.6	+	+			+	+			+	+	+	+	+	+	+			+	+	+	+	+					+	+	+	+	+	+	+	+	
		PA 6.12							+																												
		PA 12								+																											
		PBT	+	+	+		+	+			+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+
		PC	+	+				+			+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+
		PC/ABS	+	+			+	+			+	+	+	+		+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+
		PC/PBT	+	+			+	+			+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+
		PC/PET	+	+			+	+			+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+
		PE-HD	+	+	+	+	+	+			+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+
		PE-LD	+	+	+	+	+	+			+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+
		PETP	+	+							+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+
		PMMA	+	+							+	+		+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+
		POM	+	+	+	+	+	+			+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+
		PP	+	+	+	+	+	+			+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+
		PPE	+	+	+	+	+	+			+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+
PPSU	+	+	+	+	+	+			+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	
PS	+	+	+	+	+	+			+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	
PSU	+	+	+	+	+	+			+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	
PVAC	+	+	+	+	+	+			+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	
PVC-hard	+	+	+	+	+	+			+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	
PVC-soft	+	+	+	+	+	+			+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	
SAN	+	+	+	+	+	+			+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	+	

+ = Good adhesion
 - = Poor adhesion
 o = No adhesion



The feeding of the different melts can cause further defects, such as:

- ▶ Bond strength (adhesion) between first and second component
- ▶ Incipient fusion of the first component
- ▶ Warpage

Bond strength (adhesion) between first and second component

If a transparent/translucent thermoplastic component is injected, lack of adhesion can also be visible in the form of a crack, due to refractance. In many cases a partial lack of adhesion is caused by oil or grease which "crawl" into the cavity due to the mould temperature or mechanical stress.

Warpage in the soft component

The determination of the effective holding pressure time by measuring the weight of the parts is hardly possible in the case of soft TPEs, since they can be strongly compressed. Warpage may occur especially in the case of soft materials because of the compression. This is why the holding pressure parameters should be set as low as possible.

Warpage (Bimetal effect for hard/hard)

Due to the staggered injection of the melts, the second melt has a higher shrinkage potential than the first. Especially in the case of hard/hard combinations, these shrinkage differences often cause warpage.

2.23 Multi-component technology

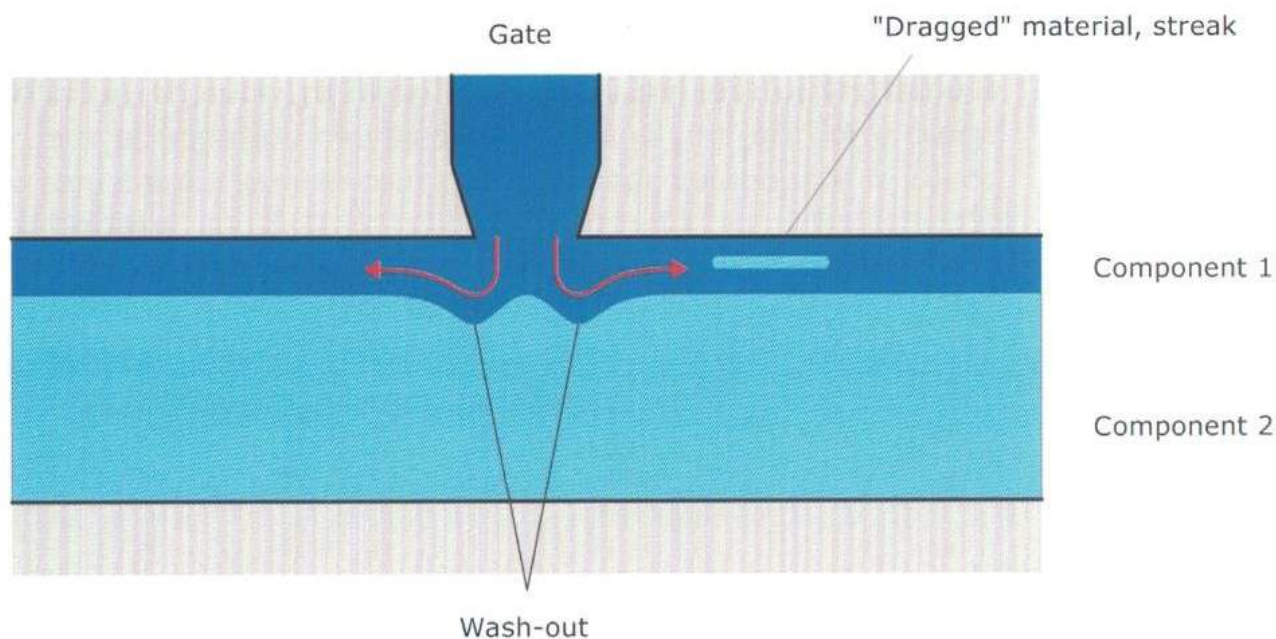
Defect described on page 40



Incipient fusion of the first component

If the gating of the second component is designed in such a way that the second melt meets the first melt at a 90° angle, melting of the first component in the gate area is possible if the melting point or the glass transition temperature are exceeded.

In the case of dyed transparent/translucent materials, this material wash-out could become visible in the form of a wall thickness variation or a coloured streak.



23.1

Incipient fusion of the
first component



Investigation

Check and/or change machine settings, change mould or moulding compound, start new cycle and go through the diagram again.

Is there poor adhesion?

NO

1. Check contact surface for contamination (oil/grease) and clean mould if necessary.
2. Reduce cycle time
3. Increase mould wall temperature of 1st component
4. Vary melt temperature of 2nd component (+)
5. Vary holding pressure time, thermoplastics (+), TPE (-)
6. Decrease mould wall temperature of 2nd component for TPE
7. Improve design of contact surfaces *1

Is the 1st component subject to incipient fusion?

NO

1. Reduce injection speed of 2nd component or injection profile (fast - slow)
2. Reduce mould wall temperature of 1st component
3. Use material with high heat resistance for the first component
4. Use film or diaphragm gate
5. Optimise mould cooling in gate area and/or gating position of 2nd component *2

Is there warpage?

NO

1st Component

1. Reduce level of holding pressure
2. Reduce holding pressure time (graded holding pressure profile)
3. Increase mould wall temperature
4. Pre-heat insert, if applicable

2nd Component

1. Increase level of holding pressure
2. Optimise holding pressure time
3. Reduce mould temperature
4. Add gate to compensate for shrinkage, if applicable

Warpage or distortion of the soft component near the gate?

1. Reduce holding pressure (falling holding pressure profile)
2. Reduce holding pressure time

*1

Increase contact surface area for hard/hard combinations (shaft, etc.) without generating sharp edges. In the case of hard/hard combinations favour blunt polished contact surfaces.

*2

Gating position of 2nd component should not be identical to gating position of 1st component. Specific temperature control too cold for 1st component near gating position of 2nd component.

2.24 Tiger lines

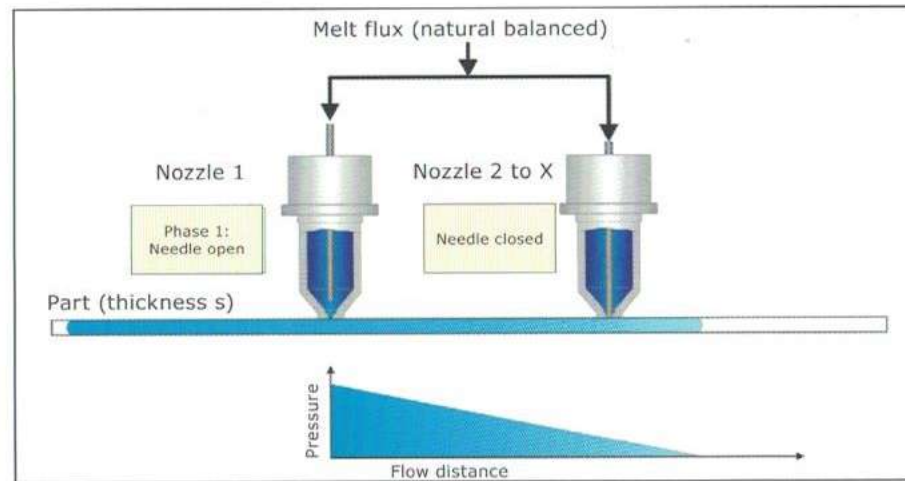
Defect described on page 41



Physical cause

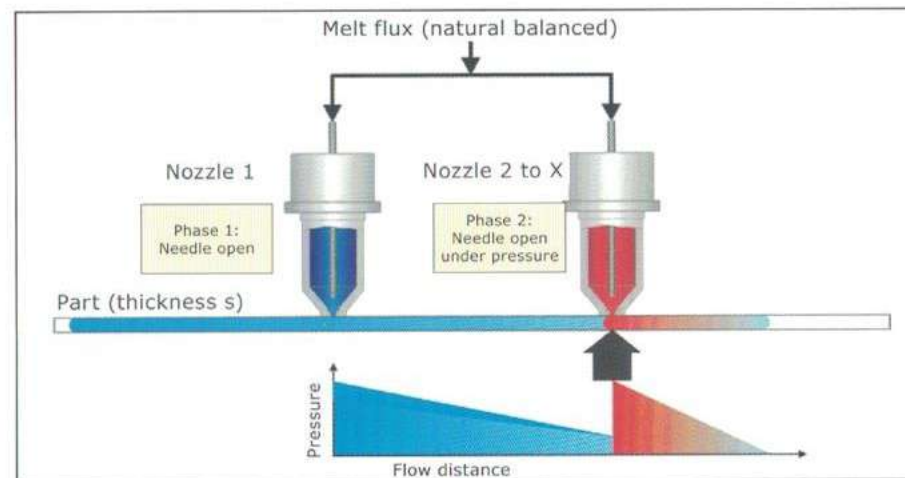
Tiger lines are alternating gloss differences (dull – shiny) on moulded parts. This surface defect only occurs with the use of specific materials. These materials are often blends (e.g. PA/ABS, PC/ PET) or thermoplastic multi-phase systems, such as PP + EPDM.

It is assumed that the differences in the reproduction of surface details and the resulting alternating gloss impressions are caused by a partial onset of crystallisation of the peripheral layer in conjunction with high shear stress. Another influencing factor is the elasticity of the melt. However, the actual cause for this defect is to be found in the material and changing processing parameters can only have a minor influence.



24.1

Pressure profile in
the case of an open
nozzle



24.2

Pressure profile after
opening another
nozzle

2.24 Tiger lines

Defect described on page 41



A reduction of the injection rate has the largest influence on the defect. Increasing the melt temperature or mould wall temperature can help indirectly, since these measures help in reducing the injection rate. In some cases, however, it may be necessary to drastically increase the injection rate.

In the case of cascade gating the pressure conditions change as another nozzle

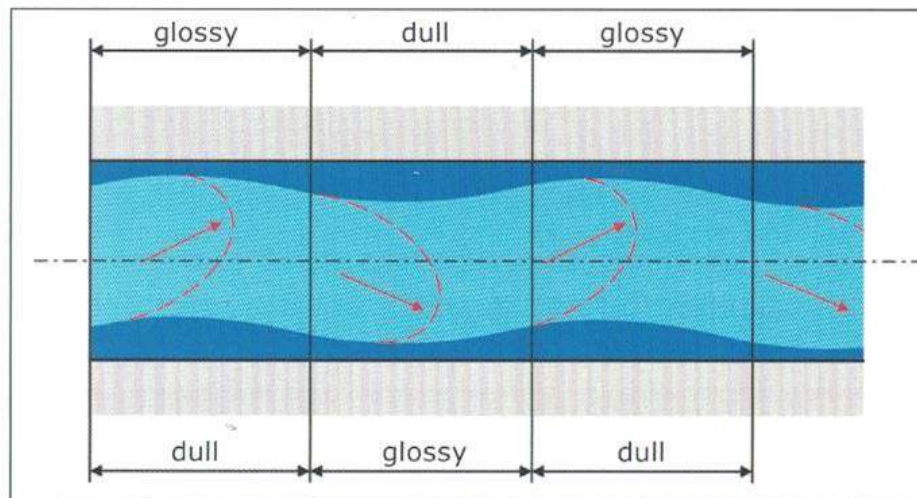
opens (see figures 24.1 and 24.2). This can increase the melt front velocity and cause tiger lines, which may only be visible in some areas of the component.

In the case of cascade gating, the use of velocity- and path-dependent needle valve opening control systems can prevent the uncontrolled entry of the melt into the cavity. As a result, abrupt changes of the melt front velocity can also be prevented.

Systems that are equipped with a pressure control system for the individual nozzles in a cascade system additionally enable constant pressure when opening the gates and, thereby, a controlled melt flow in the cavity regardless of the injection moulding machine.

Note:

Similar looking defects can also be caused by melt inhomogeneities. These can usually be optimised by changing the plasticising parameters, by using other screw geometries or by using additional homogenizing sections.



24.3

Alternating or pulsating melt centre

2.24 Tiger lines

Defect described on page 41



Investigation

Check and/or change machine settings, change mould or moulding compound, start new cycle and go through the diagram again.

Do tiger lines appear all over the surface of the moulded part?

no

1. Check for possible melt inhomogeneities *1
2. Reduce injection rate
3. Increase melt temperature
4. Increase mould wall temperature *3
5. Increase sprue and/or runner cross-section
6. Increase wall thickness
7. Use different material *2

Do tiger lines appear in certain areas of the moulded part only?

no

1. Change pressure conditions so that the flow front velocity in the flawed area of the gate is reduced
2. Reduce injection rate per nozzle
3. Increase melt temperature
4. Increase mould wall temperature *3
5. (Increase sprue and/or runner cross-section
6. Increase wall thickness
7. Use different material *2

Is a cascade being used?

1. Reduce injection rate/adapt injection profile
2. Increase melt temperature
3. Check for possible melt inhomogeneities *1
4. Use speed- and distance-dependent needle control
5. Use melt pressure control in cascade system

*1

See section 2.4, "Coloured streaks".

*2

Highest possible melt flow index, smallest possible molecular weight distribution, highest possible share of soft phases, e.g. for PP-EPDM.

*3

Raising the mould wall temperature increases the cooling time by 20 % per 10° C.



Physical causes

Inflation/bulging

Inflation of the moulded parts is caused by a peripheral layer of the part that is still, or once again, unstable, and by the internal gas pressure. If inflation occurs directly or shortly after demoulding, the peripheral layer did not have sufficient time for "growing" as a result of the solidification or for building up a sufficient level of strength in order to withstand the internal gas pressure. Inflation that occurs after a brief waiting time combined with the use of an exothermic blowing agent hints at a continuing development of heat in the demoulded part, which caused the peripheral layer to soften. In the case of very thick-walled parts, the effect may also occur irrespective of the blowing agent that is used. All of this is due to the fact that the core is still molten when the part is demoulded. As a result of the large amount of heat that is stored in the core, it softens the peripheral layer. Since the insufficient rigidity of

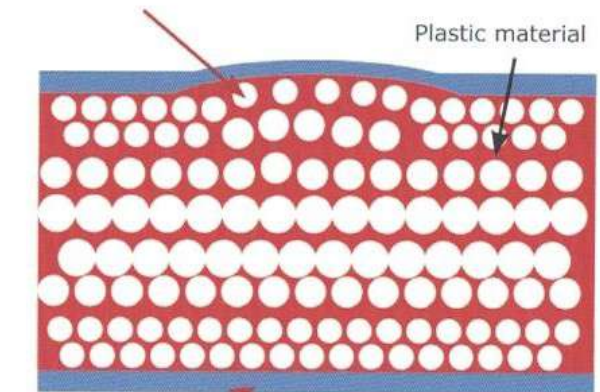
the peripheral layer is the reason for this defect, the solidification of the peripheral layer should be enhanced. Hot spots in the mould (e.g. in rib areas) should be absolutely avoided since these areas are particularly susceptible to inflation.

Discolouration

The use of blowing agents can always cause a change in colour compared to compact materials. This is mainly due to the addition of a carrier material and active agent. In addition, the reaction products can also lead to colour changes (e.g. yellowing). This section, however, is about the change in colour during a production run, which points to a degradation of the material. This may occur, for example, in the case of exothermic blowing agents when the dwell time in the plasticising unit is too long. In combination with the exothermic reaction, this may lead to a strong temperature rise in the plasticising unit, thereby causing the thermal degradation of the material.

However, endothermic blowing agents can also cause a change in colour in the course of the production run. Some active agents, for instance, release water during the reaction. In the case of hygroscopic materials, and due to the amount of heat that is present, this may cause the material to degrade.

Hot spot (thinner peripheral layer)



Stable peripheral layer



25.1

Partially inflated area of the moulded part

2.25 Defects of thermoplastic foam injection moulded parts

Defect described on page 42



Plate-out/mould corrosion

Plate-out due to chemical foaming (and not due to the properties of the base polymer) is mostly caused by the release of gas from the melt front during the filling of the moulded part. The release of gas is caused by a low pressure at the melt front (atmospheric pressure) and also by an expansion of the blowing gas. This also leads to streaks on the surface of the moulded part. Apart from the blowing gas itself, other reactants of the active agent or reactants of the active agent and base polymer also escape from the melt front and attach themselves to the mould surface. Plate-out occurs predominantly at the end of the flow path or in venting areas, since these areas are particularly susceptible to the release of gas.

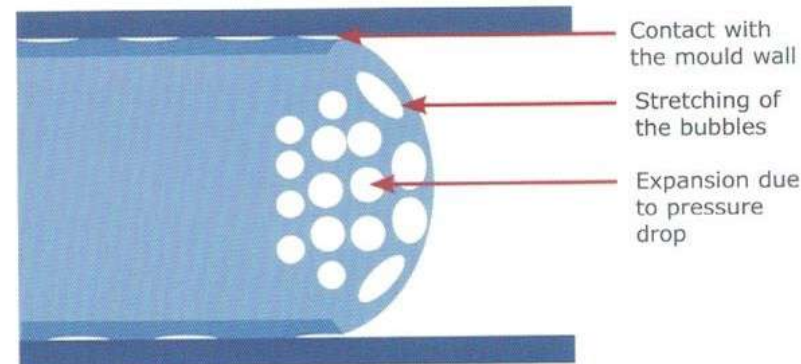
Mould corrosion is caused by active agents that release water during the

reaction or leave behind acid residues in the plastic material (e.g. citrates). This can be prevented by way of coatings or the use of corrosion-resistant steel. In addition, the acid residues can be neutralised by a suitable mixture of active agents.

Streaks/air hooks

Streaks or air hooks on the surface of the moulded part are caused by the release of gas from the melt. Thin silver streaks

that are evenly distributed over the part surface are evidence of a uniform release of gas at the melt front. Coarse air hooks and streaks indicate large gas bubbles that may be caused by expansion in the space in front of the screw or in the hot runner or by low injection rates and that stretch out along the mould wall. Due to the velocity profile on the melt front, the gas bubbles are transported to the melt front surface. Since they strive for pressure compensation, the bubbles burst,



25.2

Movement of the gas
bubbles at the melt
front



stretch out on the melt front due to the biaxial flow, and freeze on the mould wall.

Slab-like surface structure/folds

A slab-like surface can have two different physical causes. First of all, there may be melt eruptions that shoot out from the melt front and that are then enclosed in the melt that flows in. This defect can be identified very well by a cut through the moulded part, thereby revealing the presence of an internal "weld line". This is mainly due to strong loading with a blowing agent, low injection rates, and

abrupt changes of cross-sections. In addition, this defect can also be caused by a rupture of the "fresh" peripheral layer because of a too high shear stress at high injection rates that may be necessary. The still rather thin, solidified peripheral layer is stressed excessively, thereby causing it to rupture and to be pushed towards the end of the flow path (folding). Possible counter-measures are a shear reduction and the strengthening of the peripheral layer.

Sink marks

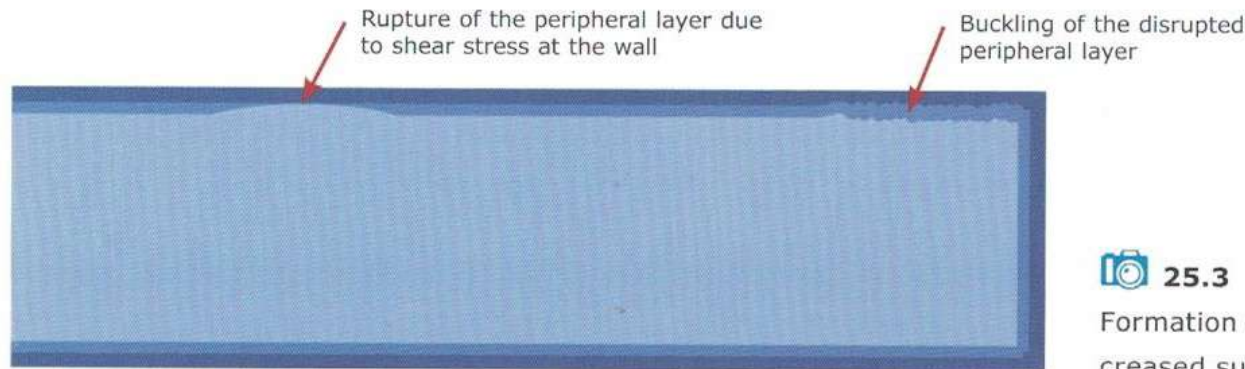
The physical cause of sink marks is an

"excessive" volumetric shrinkage of the material that cools. This shrinkage can be influenced by the process technology as well as by the part design.

If thermoplastic foam injection moulded parts (TSG parts) are equipped with thin-walled part segments, this may lead to sink marks in these areas during chemical foaming. This phenomenon is caused by insufficient gas pressures that cannot compensate for the otherwise necessary holding pressure, and also by rapid solidification in these areas.

If a holding pressure is adjusted, foaming is suppressed. Even a very short and very low holding pressure can suppress foaming to such an extent that sink marks become more distinct compared to parts without holding pressure.

In addition, sink marks can also be caused by insufficient gas pressure and volume. In this context, it must be determined as to whether the development of gas is too feeble (e.g. due to an



25.3

Formation of a creased surface

2.25 Defects of thermoplastic foam injection moulded parts

Defect described on page 42



unsuitable blowing agent, insufficient melt temperature, or insufficient dwell time) or whether the gas losses are too high (e.g. due to premature expansion, excessive melt temperature, insufficient back pressure, unsuitable blowing agent).

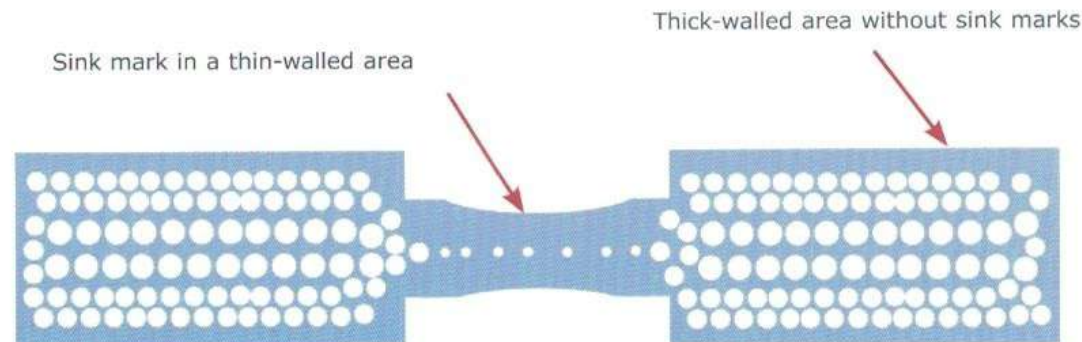
Formation of pimples

If thermoplastic foam injection moulded parts (TSG parts) receive a high-gloss finish, pimples may form after storage and the application of media (e.g. coat-

ings). These pimples are microscopic voids. This defect is caused by a reaction of residues in the part with the environment or medium. Especially endothermic blowing agents on a basis of sodium bicarbonate are known for a reaction with water. In the case of coated parts and endothermic blowing agents, citric acid derivatives should be used as the active agent.

In general, it should be taken into consideration that parts that are to be coated may show pimples and delamination

due to the release of gas. This is why a certain minimum storage period prior to coating should perhaps be planned. In addition, the blowing agent dose should be as small as possible.



25.4

Formation of sink marks in areas with abrupt changes in wall thickness



Investigation

Check and/or change machine settings, change mould or moulding compound, start new cycle and go through the diagram again.

Are there streaks on the entire surface of the moulded part?

No

1. Check material drying (measure residual moisture content)
2. Check for premature blowing agent expansion in the space in front of the screw *2
3. Increase back pressure
4. Increase injection rate
5. Reduce blowing agent dose
6. Increase mould wall temperature
7. Use blowing agent with low streaking potential
8. Optimise clamping force (+)
9. Reduce venting
10. Use insulating mould coating
11. Use special process *1

Are there streaks in areas away from the gate?

No

1. Relocate the changeover point closer to the machine nozzle or delay the changeover.
2. Optimise clamping force (+)
3. Reduce the number of venting channels *3

Are there air hooks on the moulded part?

No

1. Check for premature blowing agent expansion in the space in front of the screw *2
2. Increase back pressure
3. Increase injection rate
4. Optimise clamping force (+)
5. Reduce venting *3
6. Use special process *1

See next page

*1
Dynamic mould temperature control (heating and cooling), gas back pressure, breathing moulds.

*2
Check the position control, shut-off nozzle, and non-return valve for correct operation. Deactivate the decompression.

*3
Increase the back pressure during the injection phase. Attention: risk of diesel effect.

*4
Check for hot spots. Ensure lower temperatures for hot mould areas.

*5
Applies primarily to exothermic blowing agents.

*6
Especially in the case of flame retardants.

2.25 Defects of thermoplastic foam injection moulded parts

Defect described on page 42



Selective surface defects/no
folding?

No

1. Increase injection rate
2. Reduce blowing agent dose
3. Ensure even wall thickness transitions
4. Use gas back pressure method

Folding across the direction of
flow/slab-like surface structure?

No

1. Reduce injection rate
2. Reduce mould wall temperature
3. Use special process *1

Inflated part areas?

No

1. Check temperature homogeneity in cavity *4
2. Increase residual cooling time
3. Reduce mould wall temperature
4. Reduce blowing agent dose
5. Switch from exothermic to endothermic system

Discolouration on the part
surface?

No

1. Reduce melt temperature *5
2. Reduce dwell time
3. Increase dosing delay
4. Reduce blowing agent dose
5. Change active agent

See next page



Plate-out in the mould?

No

1. Check changeover point (reduce free foaming)
2. Check dwell time *6
3. Increase injection rate
4. Check for premature blowing agent expansion in the space in front of the screw *2
5. Reduce blowing agent dose
6. Use plate-out-minimising blowing agent
7. Plate-out minimising mould coatings
8. Check reaction of active agents with plastic components *6
9. Use gas back pressure method

Corrosion in the mould?

No

1. Change active agent
2. Use coatings
3. Use corrosion-resistant steels

Are there sink marks in thin-walled areas?

No

1. Adjust low holding pressure (check influence on thick-walled areas)
2. Increase mould temperature (perhaps only in thin-walled area)
3. Increase wall thickness

Do sink marks occur already during the start-up phase?

No

1. Check for premature blowing agent expansion in the space in front of the screw *2
2. Check material feed (e.g. for demixing)
3. Deactivate holding pressure
4. Increase back pressure
5. Increase dosing delay
6. Reduce melt cushion
7. Increase injection rate
8. Relocate the changeover point closer to the machine nozzle or delay the changeover
9. Adjust steeper temperature profile (plasticising unit)
10. Optimise melt temperature (+)
11. Increase blowing agent dose
12. Improve venting
13. Use different type of blowing agent (reaction temperature, reaction time)

See next page

CHAPTER 2

Physical causes
and remedies

2.25 Defects of thermoplastic foam injection moulded parts

Defect described on page 42



Do sink marks form during the
production run?

No

1. Check for premature blowing agent expansion in the space in front of the screw *2
2. Check material feed (e.g. for demixing)
3. Increase dosing delay
4. Reduce melt cushion
5. Increase back pressure
6. Relocate the changeover point slightly closer to the machine nozzle or delay the changeover.
7. Adjust steeper temperature profile (plasticising unit)
8. Optimise melt temperature (-)
9. Check mould venting
10. Check blowing agent for damage/ageing
11. Check non-return valve for wear

Are there pimples/bumps on the
surface?

1. Reduce blowing agent dose
2. Increase component storage time (e.g. prior to coating)
3. Check whether compact parts also show pimples/bumps => perhaps base material
4. Select another active agent (exclude reaction with medium)
5. Select another coating medium (exclude reaction)



Physical cause

Flow anomalies can be attributed to relatively intense and/or long lasting material shearing. This can cause partial advancing of the flow front, resulting in, for example, entrapped air or secondary weld lines. This advancing can occur within one cavity or among several cavities in the form of a balancing problem. In this case, natural balancing (flow paths of equal length and cross-section) of the hot and cold runner systems is insufficient.

The following parameters are common causes of material shearing:

- ▶ Insufficient melt temperature
- ▶ "Cold" areas in the hot runner system
- ▶ Small flow cross-sections
- ▶ Excessive injection rate
- ▶ Excessive duration of shear stress

During the injection phase the peripheral layers of the melt are subject to high thermal and mechanical stress inside the sprue system. Reducing the size of the flow cross-sections, the injection time and the melt temperature will cause an increase in shearing and thus frictional heat during mould filling and the viscosity of the melt will decrease. Concentration of these low-viscosity melts into one area of the mould can cause partial advancing of the flow front.

The correlation between the velocity, shear rate, temperature and viscosity profiles over half the flow cross-section are shown qualitatively in figure 26.1-4.

It is often erroneously assumed that a thermoplastic material only has one single viscosity value at a given melt temperature, also during the injection process. However, due to the flow process, a shear rate profile develops over the corresponding flow cross-section, which influences the melt temperature and thus the viscosity of the individual free-flowing layers (figure 26.1-4).

2.26 Unevenly filled parts/flow anomalies

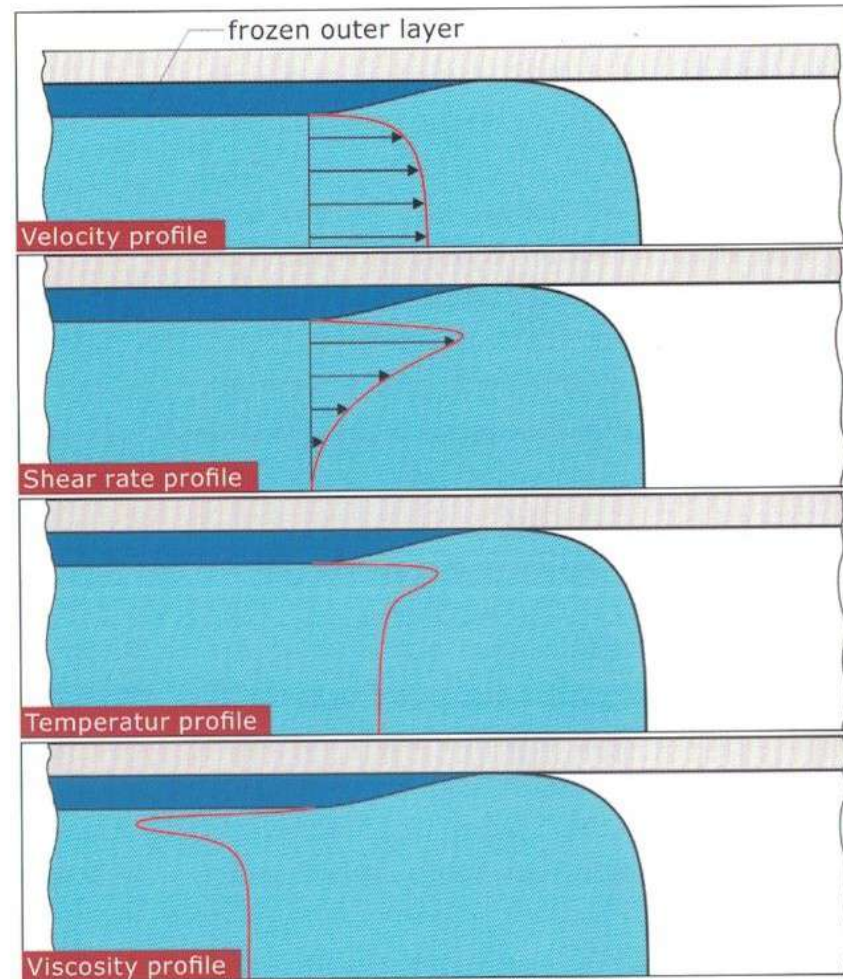
Defect described on page 47



In small flow cross-sections, these differences in viscosity may exceed a factor of 100. Thus each flow process shows a very inhomogeneous melt regarding the viscosity.

If and how a viscosity homogenisation of the melt during its flow into the cavity takes place, depends on the geometric design of the sprue systems and on the gating position at the moulded part.

Another factor influencing the viscosity homogeneity is the melt temperature in the melt plasticisation and hot runner system. The shearing duration should also be reduced in order to reduce the flow anomalies.

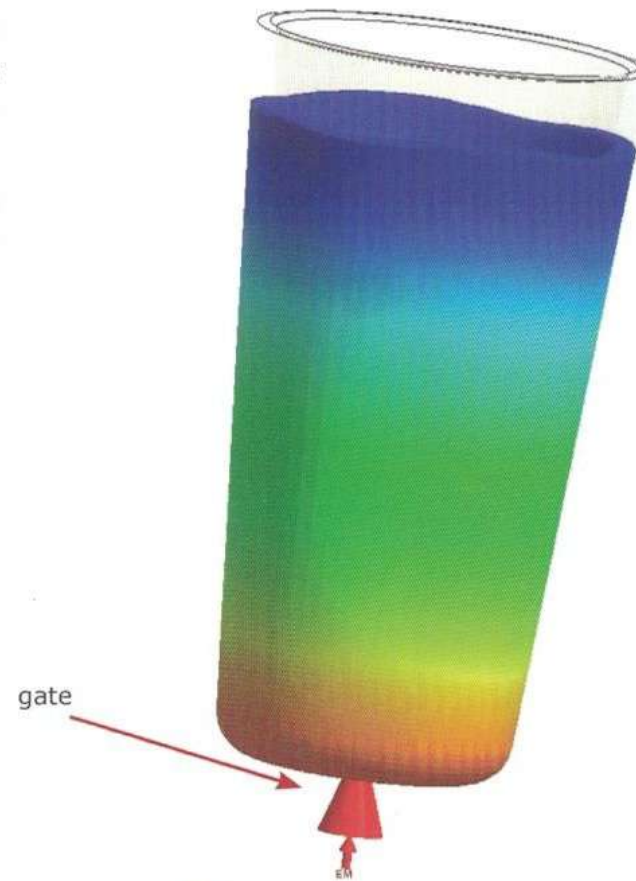


26.1 - 4

Velocity, shear rate,
temperature and vis-
cosity profile over a
half flow crosssection
(qualitative)

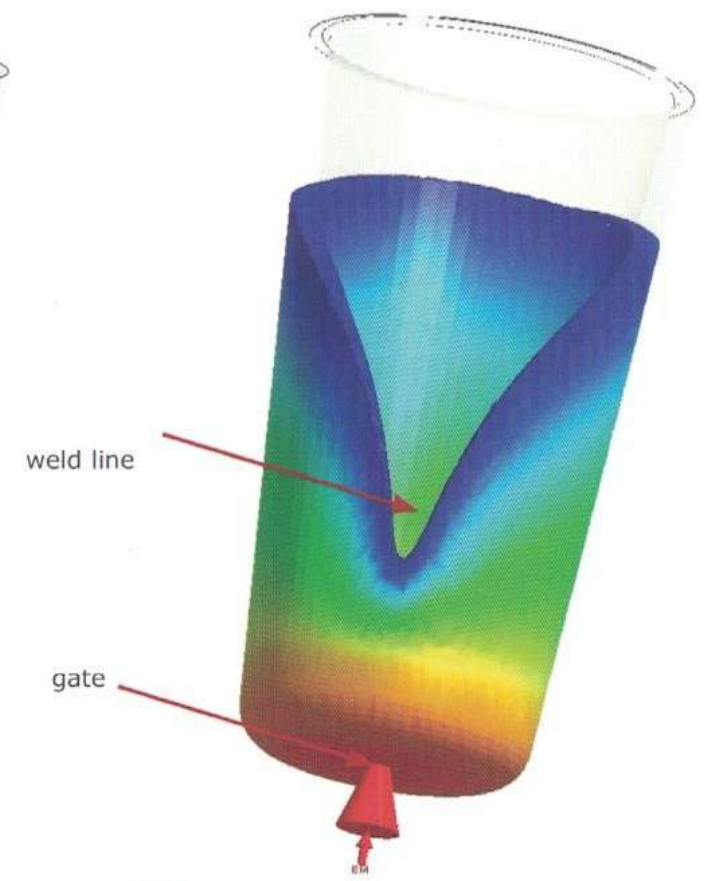


Rotation-symmetric parts that are moulded in a one-sided or unsymmetrical manner can lead to a core displacement in the mould because of partial melt front advancing effects. This leads to an uneven wall thickness distribution. This, in turn, intensifies the advancing effects so that air entrapments, weld lines, or diesel effects may occur.



 **26.5**

Even melt front progress without core displacement



 **26.6**

Uneven melt front progress with core displacement

CHAPTER 2

Physical causes
and remedies

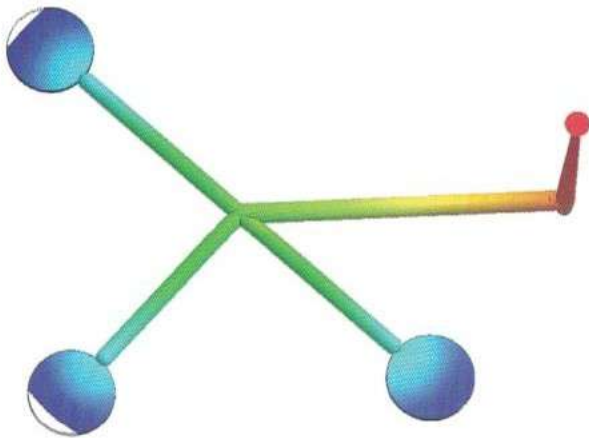
2.26 Unevenly filled parts/flow anomalies

Defect described on page 47

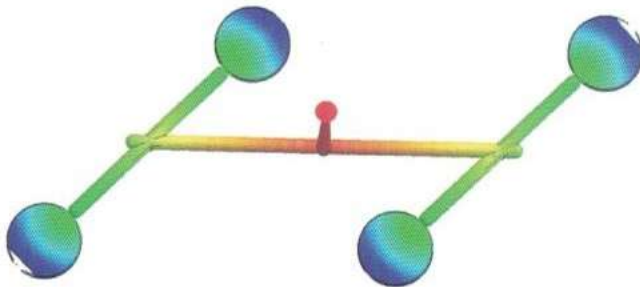


Figures 26.7 and 26.8 show different types of naturally balanced cold runner systems. Although the flow paths are of the same length, the points of time of filling differ between the cavi-

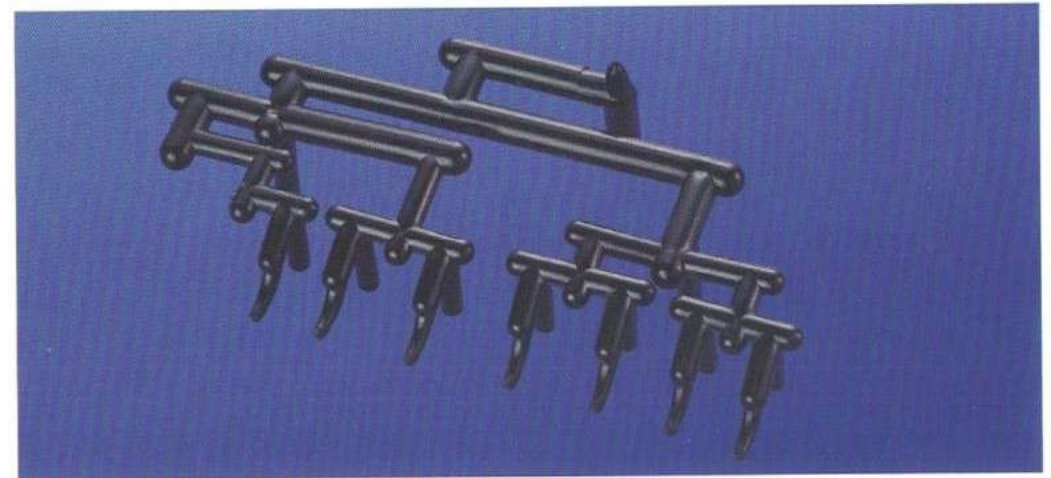
ties due to shear effects. This phenomenon occurs mainly when the main distributor and subdistributor are not perpendicular to each other.



 26.7
Y-distributor



 26.8
H-distributor



 26.9
Unbalanced runner
system



Investigation

Check and/or change machine settings, change mould or moulding compound, start new cycle and go through the diagram again.

*1
Adapt the axial temperature profile of the melt over the dosing stroke by way of the parameters cylinder temperature, back pressure, and screw speed.

Is there a hot runner?

No

1. Reduce injection rate
2. Increase melt temperature
3. Check thermal melt homogeneity *1
4. Check cross-section of gates
5. Check flow cross-section of bores
6. Check temperature homogeneity (measurement when installed)
7. Use only right-angled melt diversions
8. Turn torpedo in hot-runner nozzle by 90°

Is there a cold runner?

No

1. Reduce injection rate
2. Increase melt temperature
3. Check thermal melt homogeneity *1
4. Check cross-section of gates
5. Check cross-section for sufficient dimensions
6. Use circular (flow-facilitating) cross-sections
7. Keep length as short as possible
8. Use only right-angled melt diversions

Is the cold runner gated via an auxiliary fillet?

No

1. Choose circular (favourable for flow) runner geometry

Is there a cold runner gating?

No

1. Choose circular (favourable for flow) runner geometry

Core displacement?

1. Centre and hold mould cores in opposite part of mould
2. Optimise filling
3. More stable mould core design
4. Move injection point

2.27 Dimensional and weight variations

Defect described on page 49



Physical cause

Melt cushion variation

The non-return valve is a design element in the front area of the screw which prevents the plasticized melt from flowing back during the injection and holding pressure phase. It works best if it is closed as soon as possible. To do so, the non-return valve must be moved axially in the direction of the screw flights. During metering it moves in the opposite direction.

The aggregate can be checked for wear by setting a very long and uniform holding pressure and monitoring the screw position. If the screw does not come to a standstill during the holding pressure phase, then either the cylinder is washed out or the nonreturn valve is worn out. For further inspection, the metering stroke must be moved to another area and the process described above must be repeated.

If the screw still does not come to a standstill during the holding pressure time, the non-return valve is broken. To begin with, the newly set changeover point should be checked.

Apart from design characteristics or different functional principles, process and material related aspects also influence the closing behaviour of the non-return valve. This mostly concerns the following issues:

- ▶ The first level of the injection rate
- ▶ The decompression volume
- ▶ The available injection stroke
- ▶ The material viscosity

Low injection rates can lead to different closing speeds of the non-return valve. Over the course of several injection cycles, a varying material volume flows back into the screw flights, which can become evident in the form of variations of the residual melt cushion.

At insufficient decompression there is always a risk that the non-return valve is moved back randomly by the pressure in the space in front of the screw. Sufficient decompression is therefore important for the opening behaviour of the non-return valve.



Very short injection or metering strokes, resulting, for example, from unfavourable shot weight to screw diameter ratios, can also negatively influence the opening and closing behaviour of the nonreturn valve.

High melt viscosity leads to faster closing of the non-return valve, whereas lower melt viscosities cause a larger leakage flow and thus a possible negative impact on the closing behaviour.

Holding pressure

The effective holding pressure time can be determined by measuring the weight of different moulded parts, produced at an increasing holding pressure time. The holding pressure level should be kept constant. As soon as the weight stops increasing significantly, the effective holding pressure time is reached. This ensures that no melt flows back in an undefined manner from the moulded part into the gating system or the space in front of the screw, which could result in deviations in weight and dimensions.

2.27 Dimensional and weight variations

Defect described on page 49



Investigation

Check and/or change machine settings, change mould or moulding compound, start new cycle and go through the diagram again.

Is the residual melt cushion too small (2 - 5 mm)? *1

1. Increase metering stroke

Does the residual melt cushion fluctuate?

1. Check non-return valve and cylinder for wear
2. Check closing behaviour of non-return valve *2
3. Check decompression
4. Check material feed
5. Check shot weight/screw diameter ratio *3
6. Use non-return valve with different functional principle

Is the holding pressure set correctly?

1. Determine gate sealing point (holding pressure time)
2. Determine level of holding pressure investigation

Has the mould wall temperature changed?

1. Check and readjust mould wall temperature
2. Check cooling time *4

Has the cooling time changed?

1. Check cooling time

See next page

*1

If necessary, check the residual melt cushion at the aggregate, since some machine controls will not show the smallest screw position, but the screw stroke at the end of the holding pressure time.

*2

Increase initial injection rate.

*3

With very short injection strokes, there may be a negative impact on the non-return valve.

*4

Increasing the cooling time results in higher processing shrinkage => higher dimensional accuracy.



Has the injection time been changed?

No

1. Determine gate sealing point (holding pressure time)
2. Determine level of holding pressure
3. Check weight

Has the melt temperature changed?

No

1. Determine gate sealing point (holding pressure time)
2. Determine level of holding pressure

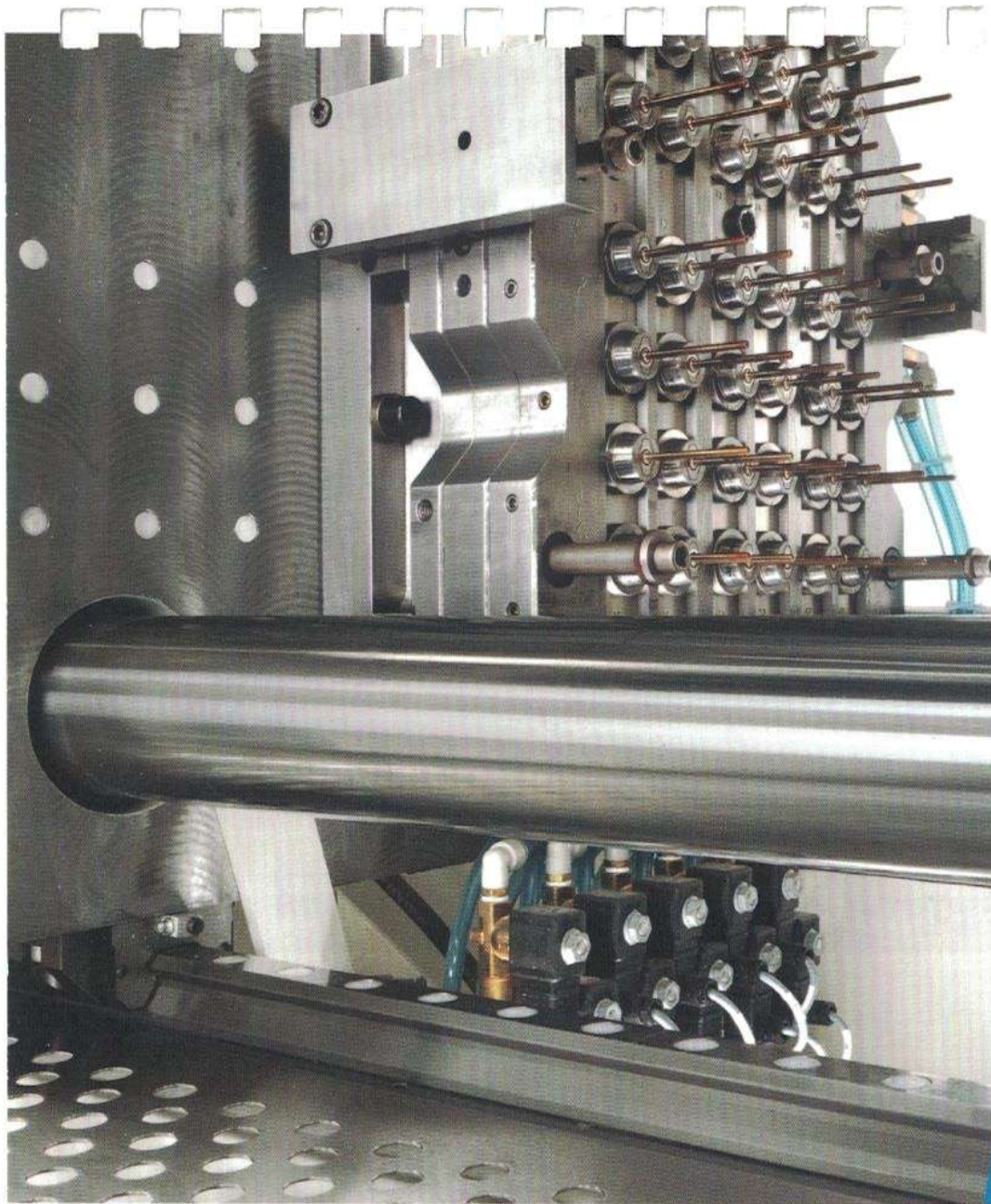
Has the batch and/or colour been changed?

No

1. Check changeover point
2. Check holding pressure level and time

1. Check sprue system for correct dimensioning
2. Check moulded part for partial wall thickness reductions
3. Check mould temperature control





Chapter 3

Effect of parameter changes on the process and on the quality of the moulded part

CHAPTER 3

Effect of parameter changes on the process and on the quality of the moulded part

3. Effect of parameter changes on the process and on the quality of the moulded part

Measure: Increasing the melt temperature

Effect on the process

Shrinkage	▲ ▼
Flowability *1	▲
Injection pressure	▼
Achievable flow path length	▲
Thermal material stress *2	▲
Holding pressure time *3	▲
Cooling time *4	▲

Effect on the moulded part

Visibility of weld lines *5	▼
Degree of orientation *6	▼
Degree of crystallisation	▲

*1 The melt temperature has a much stronger influence on the viscosity of amorphous thermoplastics than on the viscosity of partially crystalline thermoplastics.

*2 In the case of thermoplastic materials and, in particular, in combination with a long dwell time of the melt in the plasticising cylinder, an increase of the melt temperature can lead to thermal material damage.

*3 See the "holding pressure time" parameter

*4 In general, the influence of the temperature increase of the melt on the cooling time can be neglected.

*5 The visible weld line notch will be levelled out to a higher extent.

*6 Highly oriented, amorphous materials (PC, PS, PMMA, ABS etc.) tend to a stronger resetting movement of the molecular chains (shrinkage). In highly oriented parts, shrinkage occurs at lower temperatures, while it occurs only at higher temperatures in the case of parts with a low level of orientation.

3. Effect of parameter changes on the process and on the quality of the moulded part

Measure: Increasing the mould wall temperature while maintaining the demoulding temperature

Effect on the process		Effect on the moulded part	
Moulding shrinkage	▲	Dimensions	▼
After-shrinkage	▼	Tolerance fluctuation	▼
Achievable flow path length	▲	Visibility of weld lines ^{*3}	▼
Holding pressure time ^{*1}	▲	Surface moulding ^{*4}	▲
Cooling time ^{*2}	▲	Degree of orientation ^{*5}	▼
		Crystallinity	▲
		Internal stress	▼

*1 See the "holding pressure time" parameter

*2 An increase of the mould wall temperature extends the cooling time by approximately 20% for every 10°C.

*3 The visible weld line notch will be levelled out to a higher extent.

*4 High-gloss polished surfaces will become glossier. Structured surfaces will become duller.

*5 Highly oriented, amorphous materials (PC, PS, PMMA, ABS etc.) tend to a stronger resetting movement of the molecular chains (shrinkage). Shrinkage occurs in highly oriented parts, while it occurs only at higher temperatures in the case of parts with a low level of orientation.

Note:

A reduction of the mould wall temperature, e.g. in order to reduce the cooling time, involves the risk that some part properties may be negatively affected which could remain unnoticed. This applies particularly to changes of the internal component properties that cannot always be detected after the manufacturing process. After-shrinkage or a loss of orientation, for example, can lead to unwanted shape and/or dimensional changes at a later point of time.

CHAPTER 3

Effect of parameter changes on the process and on the quality of the moulded part

3. Effect of parameter changes on the process and on the quality of the moulded part

Measure: Increasing the holding pressure

Effect on the process		Effect on the moulded part	
Shrinkage	▼	Dimensions	▲
Clamping force	▲	Weight	▲
Demoulding problems *1	▲	Sink marks	▼
Mould deformation	▲	Formation of voids	▼
		Weld line stability	▲
		Surface moulding	▲
		Flashing	▲
		Degree of orientation	▲

*1 In the case of thin-walled and/or ribbed parts, a high holding pressure can lead to demoulding problems, since these part areas cannot sufficiently shrink away from the mould wall.

3. Effect of parameter changes on the process and on the quality of the moulded part

Measure: Increasing the injection rate

Effect on the process		Effect on the moulded part	
Thermal material stress *1	▲	Dull spot in the gate area	▲
Shearing *1	▲	Jetting	▲
Venting problems	▲	Air streaks, air hooks	▲
Achievable flow path length	▲	Flashing	▲
Injection time	▼	Glass fibre streaks	▼
		Burnt streaks	▲
		Diesel effect	▲
		Surface moulding	▲
		Record grooves effect	▼

*1 High injection rates lead to a high level of shear in the melt, which in turn may lead to thermal damage. Under high shear stress combined with long dwell times, shear-sensitive materials, e.g. ABS, PMMA, or PC, tend to undergo partial material degradation resulting in streaks on the part surface, for example.

CHAPTER 3

Effect of parameter changes on the process and on the quality of the moulded part

3. Effect of parameter changes on the process and on the quality of the moulded part

Measure: Increasing the back pressure

Effect on the process		Effect on the moulded part	
Melt homogeneity *1	▲	Coloured streaks *1	▼
Melt temperature *2	▲	Stability *4	▼
Flowability *3	▲	Air streaks *6	▼
Glass fibre length *4	▼		
Dosing time	▲		
Dosing time fluctuations *5	▼ ▲		

*1 A higher back pressure improves the thermal and mechanical melt homogeneity. Coloured streaks will be reduced.

*2 See the "melt temperature" parameter

*3 Higher back pressures generate a high level of friction (frictional heat) in the melt. The frictional heat reduces the viscosity of the melt and improves its flowability.

*4 In the case of plastics that are reinforced with long glass fibres, a high back pressure leads to strong damage to the glass fibres and, thereby, to a reduction of the mechanical component properties.

*5 In the case of plastic materials with internal lubricants or colour batches, back pressures that are too high or too low can lead to dosing fluctuations.

*6 The back pressure pushes any air that has been drawn in together with the granules in the direction of the feeding opening, thereby preventing the air from entering the space in front of the screw.

3. Effect of parameter changes on the process and on the quality of the moulded part

Measure: Increasing the screw speed

Effect on the process		Effect on the moulded part	
Melt homogeneity *1	▼	Coloured streaks *6	▲
Melt temperature*2	▲	Stability *7	▼
Flowability *3	▲		
Thermal material stress *4	▲		
Glass fibre length *7	▼		
Uneven mould filling *5	▲		
Dosing time *1	▼		

*1 Higher screw speeds reduce the dosing time. Very short dosing times lead to thermal and mechanical inhomogeneities in the melt.

*2 See the "melt temperature" parameter

*3 Higher screw speeds generate a high level of friction (frictional heat) in the melt. The frictional heat reduces the viscosity of the melt and improves its flowability.

*4 High melt temperatures – caused by frictional heat – can lead to thermal material damage.

*5 Short dosing times in combination with high dosing volumes lead to high axial temperature differences in the space in front of the screw.

*6 If dyeing is performed by way of a master batch, high screw speeds can lead to an uneven mixing of the melt with the dye.

*7 Mechanical melt inhomogeneities, e.g. agglomeration, can reduce the strength in a component. In the case of plastics that are reinforced with long glass fibres, a high screw speed leads to strong damage to the glass fibres and, thereby, to a reduction of the mechanical component properties.





Chapter 4

General notes on the identification and correction of defects

CHAPTER 4

General notes on the identification and correction of defects



4. General notes on the identification and correction of defects

What makes the identification of defect causes so difficult is the large variety of potential influencing factors. Apart from process-related parameters, defects can also be caused by design aspects (part and mould), the material and its preparation (e.g. drying), the periphery, or the environment (cleanliness, climate), either alone or as a combination. This is aggravated by the fact that some defects have a similar outer appearance, although their physical causes can be of a completely different nature. This is why the procedure for the correction of defects can vary to a large extent. In some cases, the underlying cause can be characterised precisely by the outer appearance of the defect. In other cases, the cause can only be narrowed down by the process of elimination. The following section provides tips that may be useful for the identification of defects.

General questions for the identification of defects

- ▶ Does the occurrence of a defect correlate with some kind of change or modification (another machine, periphery, batch change, etc.)?
- ▶ Is it a systematic defect (frequency and location of occurrence)?
- ▶ Can the physical cause be identified/narrowed down based on the outer appearance of the defect?
- ▶ Does the outer appearance of the defect provide information on the phase of the injection moulding process (injection, compression, holding pressure phase, etc.) during which the defect is produced?
- ▶ Does the defect occur in the context of a process interruption?
- ▶ Is it necessary to adapt the parameters for a batch change?
- ▶ Does the defect also occur in combination with an alternative material, e.g. if a natural material is used (eliminating the influence of glass fibres and additives)?
- ▶ Does the defect also occur with another machine?
- ▶ Is the applied cooling time in line with the wall thickness of the part?
- ▶ Are there hot spots in the mould?
- ▶ To what extent can the defect be influenced by process-related optimisations? Is it necessary to make changes to the item or mould in order to realise further optimisations?

4. General notes on the identification and correction of defects

Material preparation/drying process:

- ▶ Clean the dryer, filter, and machine hopper thoroughly whenever the material is changed.
- ▶ Place a cover on the machine hopper. Use magnets in the hopper.
- ▶ Protect open bags of granules against contamination.
- ▶ Is the dryer volume adapted to the dosing volume and the resulting dwell time?
- ▶ Is the drying temperature and duration in the recommended range?
- ▶ Is the residual moisture content in the recommended range (use a suitable measurement method)?
- ▶ Is the type of dryer suitable for reaching the required residual moisture content?
- ▶ Is the correct operation of the dryer (molecular sieve) ensured?
- ▶ What happens to the material after the drying process? For example, is the material highly hygroscopic (uptake of moisture in the hopper)?

- ▶ In the case of small shot weights, adapt the material volume in the hopper in order to prevent the uptake of moisture.
- ▶ Will the hopper be closed with a cover?
- ▶ Is it possible to use a top-mounted dryer?
- ▶ Does the material take up moisture in the feed pipes?

Temperature control of the periphery

- ▶ Have the feed flow temperatures been set to the correct values at the temperature control unit?
- ▶ Is the flow rate sufficient?
- ▶ Is the power of the temperature control unit sufficient?
- ▶ Have the temperature control circuits been connected correctly?

CHAPTER 4

General notes on the identification and correction of defects



4. General notes on the identification and correction of defects

Process observation

1. Observation of the screw movements

Injection phase

- ▶ Does the machine reach the specified volumetric flow rate or injection rate?
- ▶ Does the machine reach the injection pressure limit?
- ▶ Are there strong fluctuations of the injection pressure?

Changeover point

- ▶ Are there fluctuations of the residual melt cushion?
- ▶ Is the changeover point overrun or not reached?
- ▶ Does the screw bounce back when the changeover point is reached?

Start of the holding pressure phase

- ▶ Does the screw move forward?

End of the holding pressure phase

- ▶ Does the screw bounce/spring back?
- ▶ Are there fluctuations of the residual melt cushion?
- ▶ Check the first level of the injection rate (closing characteristics of the non-return valve).
- ▶ Check the non-return valve and cylinder for signs of wear.
- ▶ Is the infeed of granules OK?
- ▶ Is the residual melt cushion in the optimum range (too small, too big)?

2. Mould movement/demoulding process

- ▶ Does the moulded part get caught on the ejector side?
- ▶ Does the moulded part get jammed during the mould opening and/or demoulding process?
- ▶ Are the gates properly demoulded?

- ▶ Do the gates tear off properly from the moulded part?
- ▶ Has the clamping force been set as high as necessary (venting)?
- ▶ Has the clamping pressure been set to a reasonable value?

3. Check of the plasticising conditions

- ▶ Is the dosing volume adapted to the screw diameter?
- ▶ Check the flange temperature.
- ▶ Adjust the back pressure correctly.
- ▶ Is the measured melt temperature in the recommended processing range?
- ▶ The melt must be injected „into the air“ with the set injection profile. In the case of small volumes, the melt cake is ideally injected into an insulated container.

4. General notes on the identification and correction of defects

- ▶ Measure the temperature with a probe thermometer. (Do not leave the melt cake on the machine base!)
- ▶ Are there dosing time fluctuations?
- ▶ Is the dosing time within the residual cooling time?
- ▶ Very short dosing times can lead to thermal and mechanical material inhomogeneities. (Check the plasticising performance of the machine.)

4. Other process fluctuations

- ▶ Adjust the decompression correctly.
- ▶ Are there deviations of the mould wall temperature during the process?

Location of appearance and formation of a defect

With some defects (e.g. streaks), the location where the defect appears and the location where the defect is formed are not identical. For example, air from unvented or poorly vented areas (ribs) can be dragged along by the melt front when the filling process continues. It then presents itself in the form of

streaks or clouds in another location. During the injection phase, it is also possible that material in the gate system is thermally damaged as a result of an excessive level of shear stress. This can lead to the formation of streaks that can be located in a different area of the part away from the gate.

Mould filling studies for identifying defects

During the analysis of potential causes of defects, partial mould filling is extremely important. Partial mould fillings show the course of the melt front as a function of the injection conditions (injection profile) and of the geometrical design of the part. Mould filling studies enable conclusions to be drawn concerning the defects and their location of origin.

In the case of multiple cavities, partial fillings also reveal uneven filling between different cavities.

The general, recommended procedure is to keep the set dosing volume constant, to retract the changeover point, and

to then change it in reasonable steps. The advantage of this method is that an injection profile that has been set in a path-dependent manner does not need to be changed. It is recommended to check the set changeover point visually with regard to the screw movement during the injection process in order to ensure that it is not overrun. It may be necessary to select very small values for the holding pressure so that the changeover point will not be overrun.

In addition, it must be ensured that no plastic material mass can enter the mould cavity during the holding pressure phase since, otherwise, the course of the melt front would be changed, resulting in incorrect results.

In order to ensure that no mass can enter the mould cavity during the dosing process, a sufficiently long dosing time delay is required.

The graduation of the partial filling steps should be particularly small in areas with changing cross-sections or areas

CHAPTER 4

General notes on the
identification and
correction of defects



4. General notes on the identification and correction of defects

showing surface defects so that the flow process can be analysed in a particularly precise manner. In addition, the partial filling specimens should be labelled with their respective changeover point or dosing stroke in order to be able to reconstruct the melt front position. For the production of partial fillings, sufficient prefilling is essential in order to reach the corresponding ejector positions.

In addition, the course and progress of the melt front must be compared between two filling levels in order to detect and identify melt standstill or atypical melt front courses in certain areas of an item (e.g. ribs).

4. General notes on the identification and correction of defects



 4.1



 4.2



 4.3



 4.4



 4.5



 4.6

 4.1 - 4.6
Filling study of a
moulded part

CHAPTER 4

General notes on the identification and correction of defects

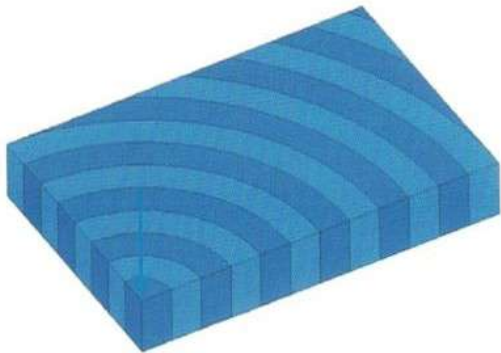
4. General notes on the identification and correction of defects

Injection phase

In order to achieve a uniform surface quality, the melt front velocity should be relatively constant. The melt front velocity is the distance that the melt front covers during the mould filling process per time unit. With a constant screw advance speed, the plastic melt

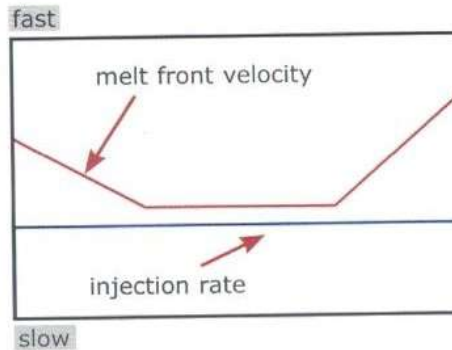
is injected with a constant volumetric flow rate. Because of the „smaller“ melt front cross-section in the areas that are close to, or far from, the gate (Figures 4.7 - 4.8 + 4.10), the melt front velocity in these areas is higher than in the area between. In order to achieve a constant melt front velocity, the injection profile

should be adjusted to „slow-quick-slow“. After the „size“ and position of the melt front cross-sections of a part, the screw advance speed must be adjusted so that the resulting melt front velocity is approximately uniform (Figures 4.9 + 4.11).



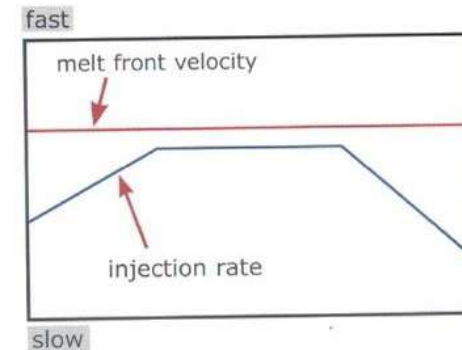
4.7

Change of the melt front cross-section over the flow path length



4.8

Course of the melt front velocity with a constant screw advance speed



4.9

Course of the melt front velocity with a staged screw advance speed

4. General notes on the identification and correction of defects

Note:

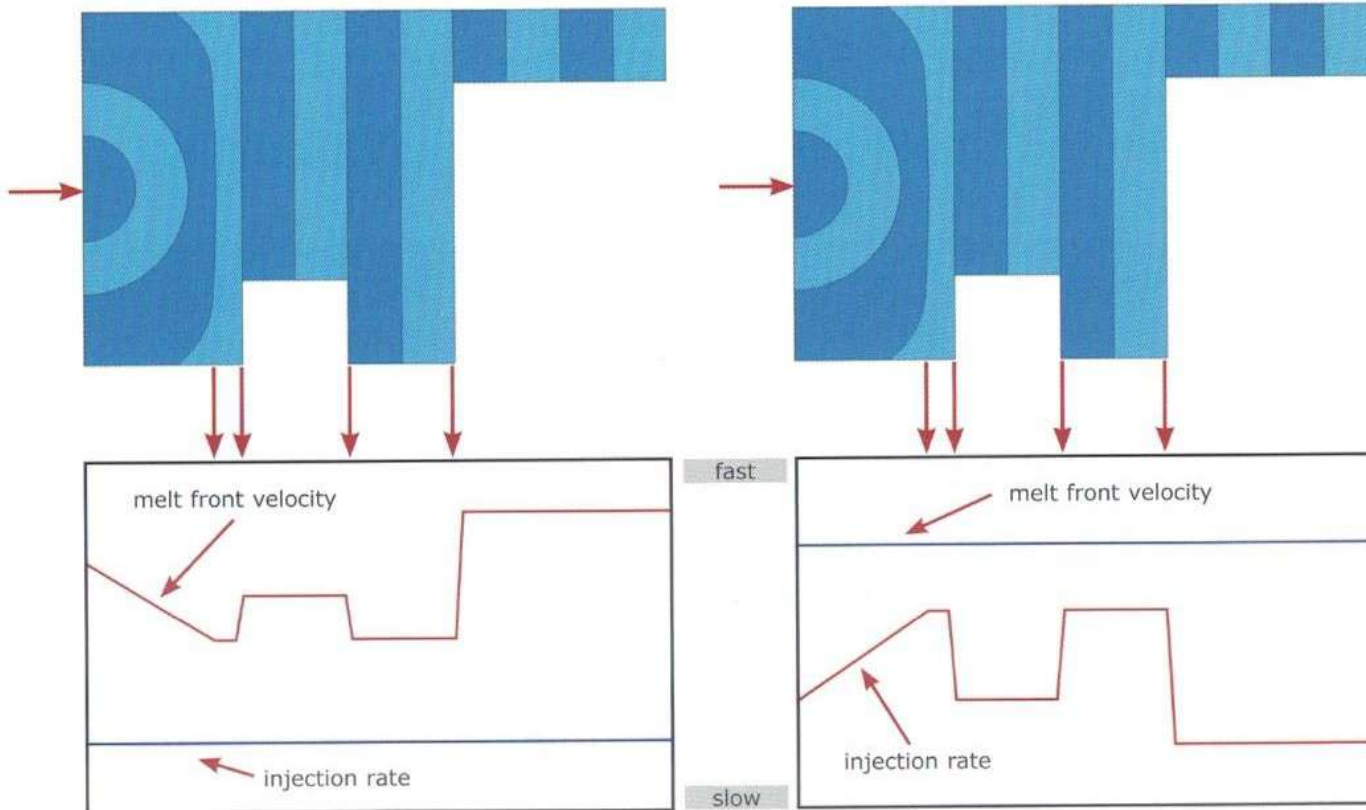
With a profiled injection rate, the following effects can be realised:

1. More precise control of the changeover point and, thereby, a more stable changeover from the injection phase to the holding pressure phase.
2. Prevention of jetting and dull spots in the gate area.
3. Prevention of surface defects, e.g. air hooks and air streaks, in poorly vented mould areas.
4. Reduction of pressure peaks at the end of the flow path.
5. Prevention of burns at the end of the flow path.
6. Prevention of burnt streaks.

CHAPTER 4

General notes on the identification and correction of defects

4. General notes on the identification and correction of defects



Note:

When the injection rate is changed, the changeover point, injection time monitoring, and pressure limitation must be corrected.

4.10

Course of the melt front velocity with a constant screw advance speed

4.11

Course of the melt front velocity with a staged screw advance speed

4. General notes on the identification and correction of defects

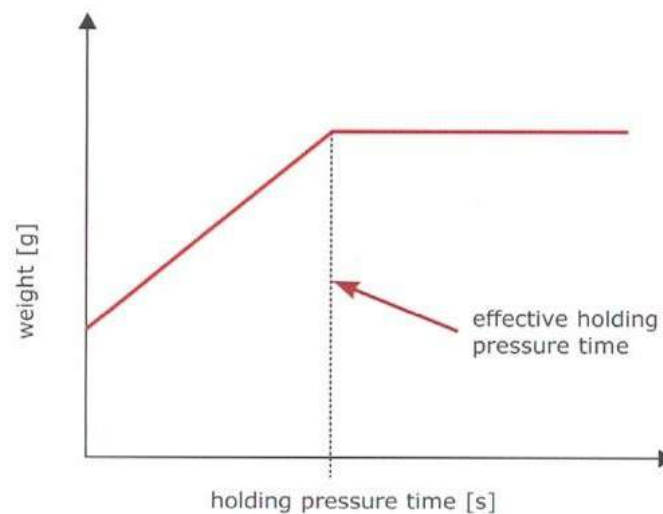
The changeover from the injection phase to the holding pressure phase is determined by the changeover point. In order to prevent the part from being overpacked, which could result in mould damage, the changeover should occur when the part is filled by approximately 98%. The changeover point needs to be checked whenever the melt volume that is to be metered is influenced by a parameter change:

- ▶ back pressure (compressibility of the melt)
- ▶ decompression (change of the plastic material volume in the space in front of the screw)
- ▶ rate of the first injection level (closing characteristics of the non-return valve)

Determination of the effective holding pressure time (sealing point)

The effective holding pressure time can be determined by measuring the weight of various parts that were produced with an increasing holding pressure time

(Figure 4.12). The holding pressure itself should be kept constant during this process. When the weight no longer increases significantly, the effective holding pressure time has been reached. This ensures that the mass cannot flow back into the gate system or into the space in front of the screw in an undefined manner, which could lead to weight and dimensional fluctuations.



Note:

This method does not apply to thermoplastic elastomers (TPE), since there would be a risk of overloading the soft component. Soft mixtures, in particular, can be compressed very strongly. This could result in surface defects (e.g. warpage of the soft component).

In the case of needle valve systems, it must be ensured that the gate has not frozen too strongly, i.e. that the needle valve can still close.

4.12

Determination of the effective holding pressure time



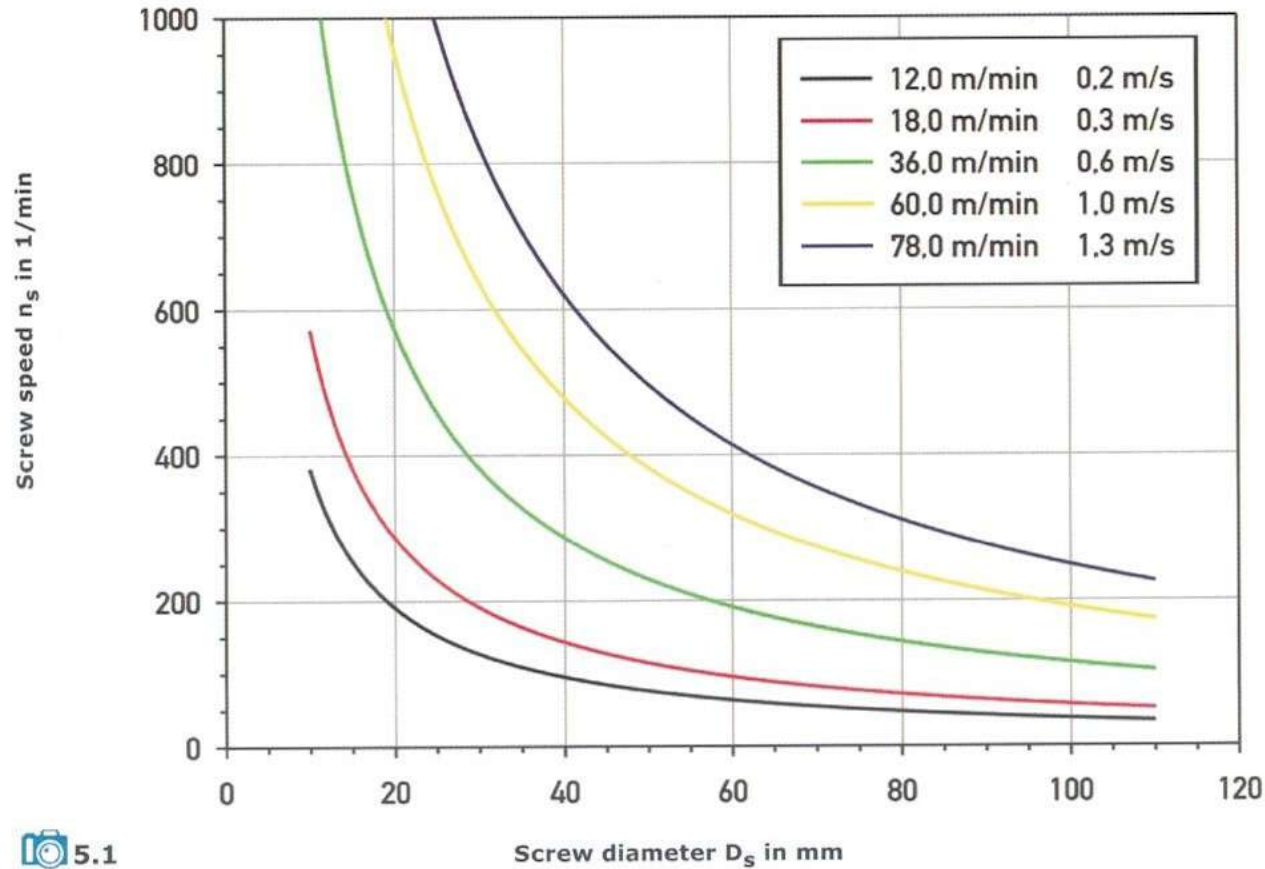


Chapter 5

General processing instructions

5. Processing instructions

Screw speed diagram



Max. peripheral speed
(m/min)

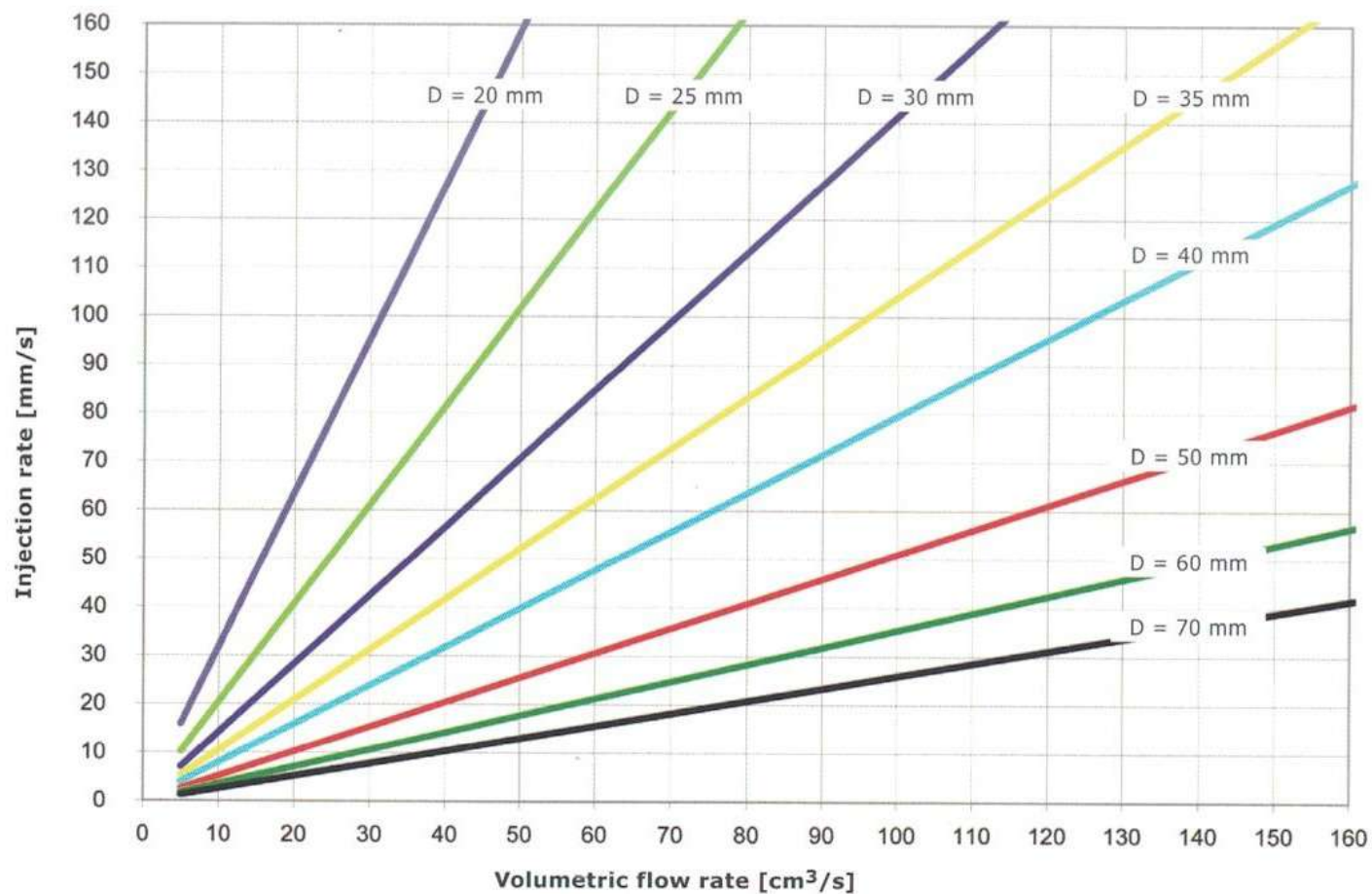
PS	60	SB	36
ABS	18	SAN	18
PMMA	18	PETP	12
PC	18	PBTP	12
PA	36		
PE-LD	60		
PE-HD	60		
PP	60		
POM	36		

Max. peripheral speed
(m/s)

PS	1	SB	0.6
ABS	0.3	SAN	0.3
PMMA	0.3	PETP	0.2
PC	0.3	PBTP	0.2
PA	0.6		
PE-LD	1		
PE-HD	1		
PP	1		
POM	0.6		

5. Processing instructions

Injection rate



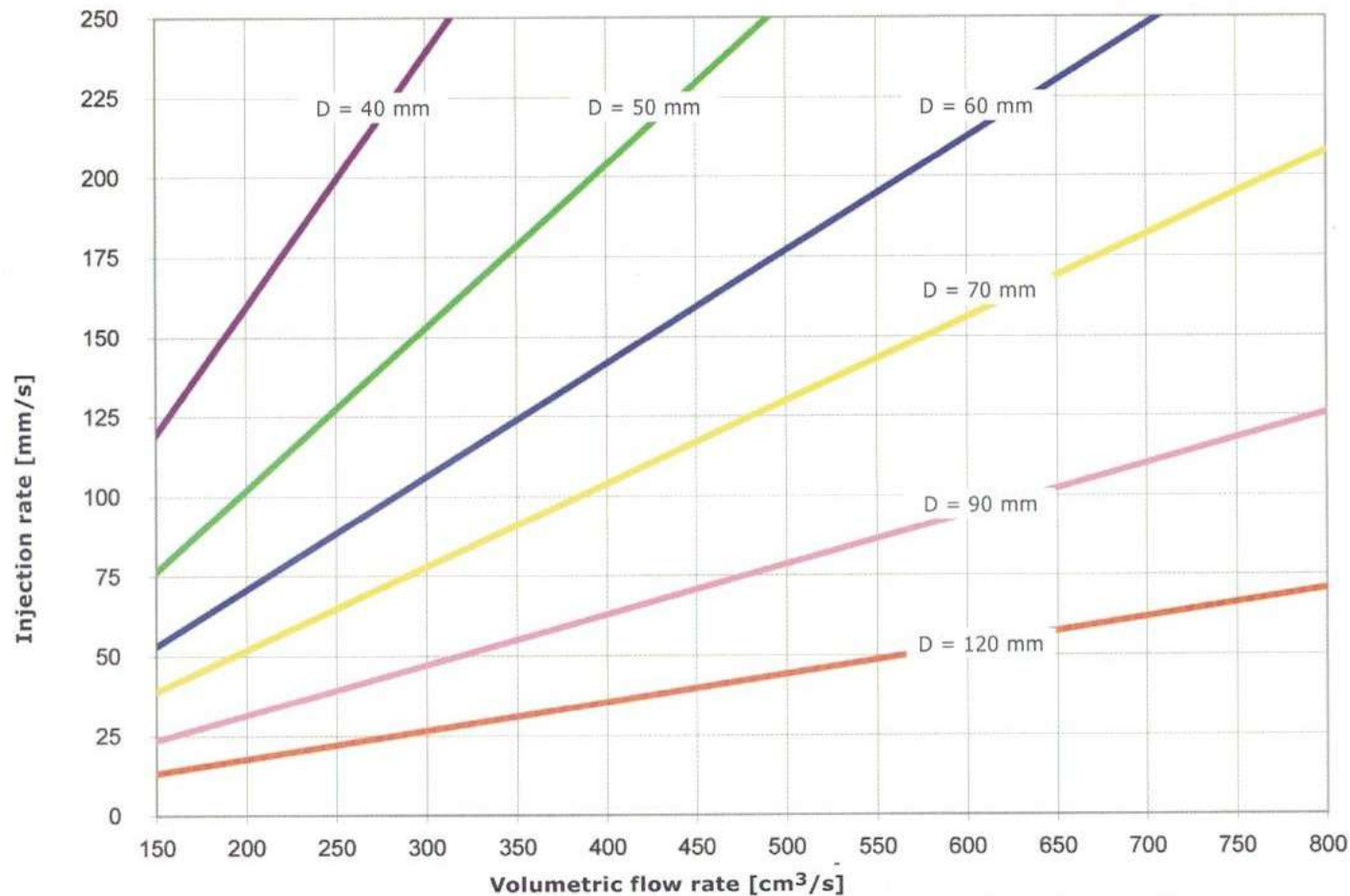
Figures 5.2 and 5.3 show the relationship between the injection rate and injection volume.

5.2

Injection rate as a function of the volumetric flow rate (up to $160 \text{ cm}^3/\text{s}$) and screw diameter $[D]$

5. Processing instructions

Injection rate

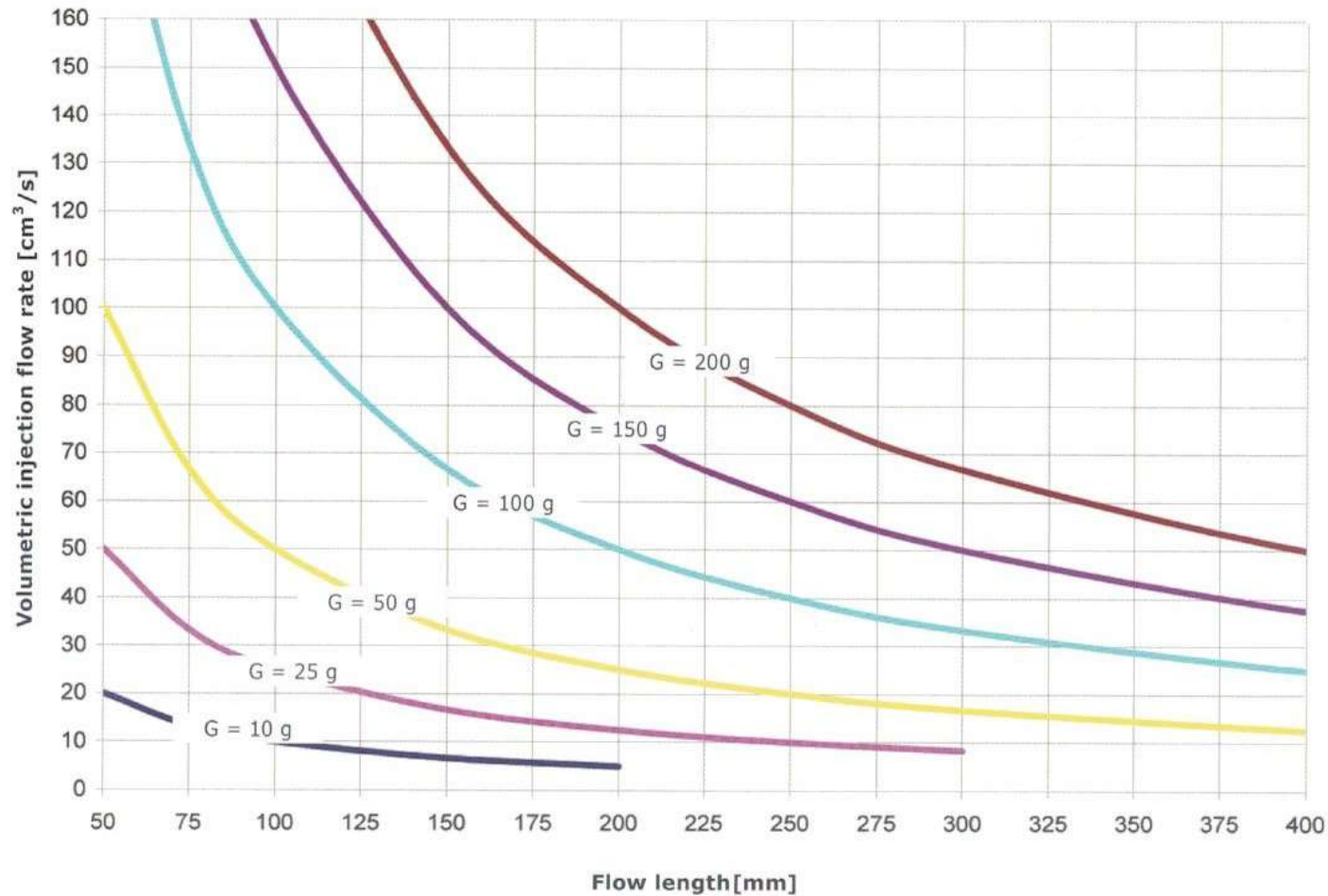


5.3

Injection rate as a function of the volumetric flow rate (150 cm³/s to 800 cm³/s) and screw diameter [D]

5. Processing instructions

Volumetric injection flow rate



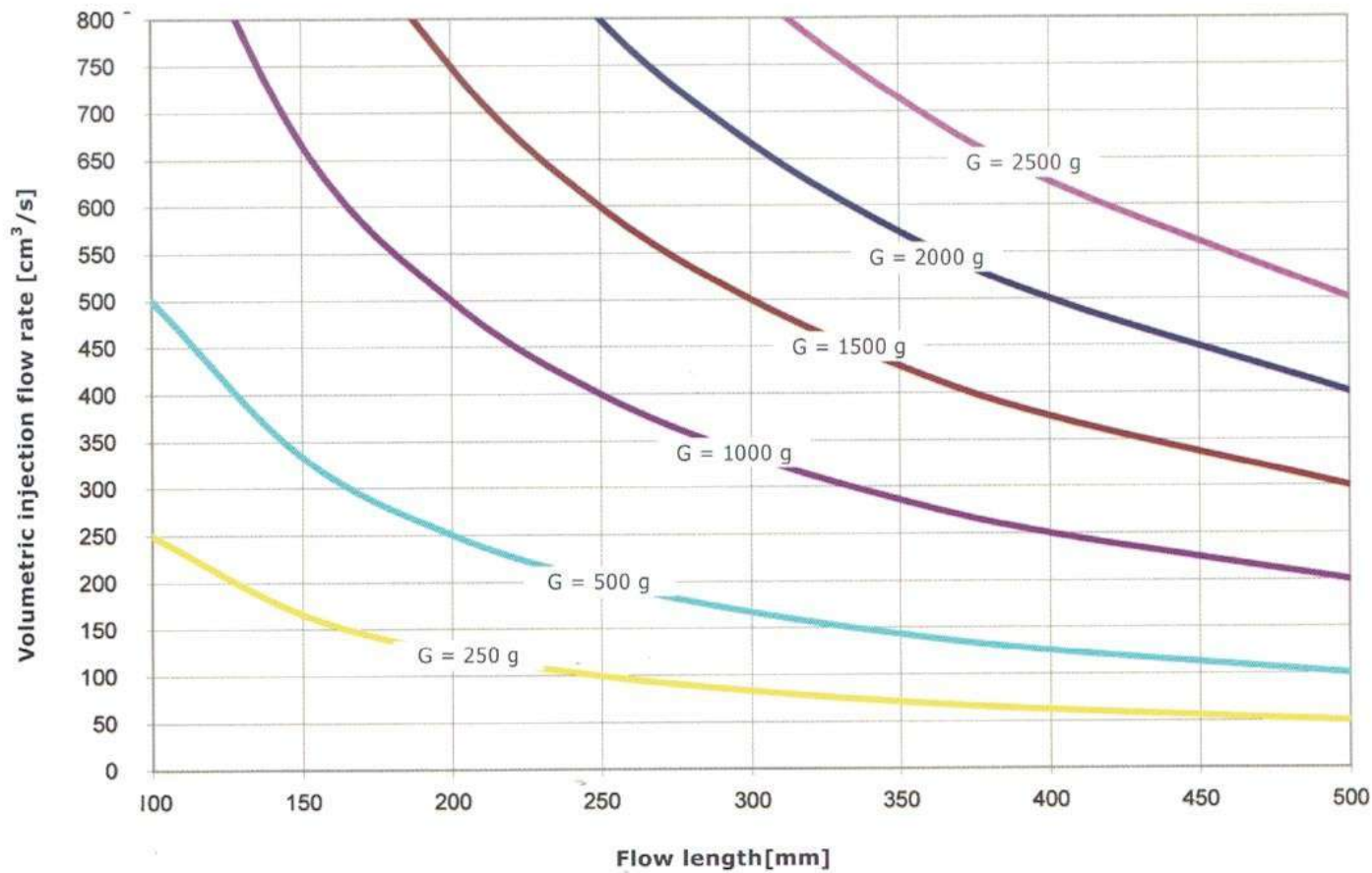
Figures 5.4 and 5.5 show the relationship between the volumetric injection flow rate and flow path length for various part weights.

5.4

Volumetric injection flow rate as a function of the part weight $[G]$ (up to 200 g) and flow path length

5. Processing instructions

Volumetric injection flow rate



5.5

Volumetric injection
flow rate as a function
of the part weight $[G]$
(250 g to 2500 g) and
flow path length

5. Processing instructions

Amorphous thermoplastics

Material	Melt temperature [°C]	Mould temperature [°C]	Flange temperature [°C]	Specific back pressure [bar]	Peripheral speed [m/min]	Peripheral speed [m/s]	Drying temperature [°C] *	Drying time [h] *
ABS	220 – 260	40 – 80	30 – 50	70 – 150	18	0.3	80	2 – 3
CA	180 – 220	50 – 80	40	80 – 100	18	0.3	60 – 65	2 – 3
CAB	180 – 220	50 – 80	40	80 – 100	36	0.6	60	2 – 3
CP	180 – 220	50 – 80	40	80 – 100	36	0.6	60	2 – 3
PA amorph	260 – 300	70 – 100	60 – 80	50 – 100	36	0.6	100 – 120	6 – 12
PC	280 – 320	80 – 120	70 – 90	50 – 150	18	0.3	120	2 – 3
PC/ABS	250 – 290	60 – 90	60 – 80	50 – 150	12	0.2	90 – 110	2 – 4
PEI	340 – 425	140 – 180	100 – 120	40 – 80	30	0.5	150	4 – 6
PES	330 – 370	100 – 150	100 – 120	30 – 80	36	0.6	190	3
PMMA	220 – 260	50 – 80	50 – 60	30 – 90	18	0.3	80	3
PPO	270 – 310	70 – 120	40	60 – 80	18	0.3	110	2
PS	190 – 280	20 – 80	20 – 30	150 – 300	60	1	80	1 – 2
PSU	310 – 390	95 – 115	100 – 120	80 – 120	36	0.6	120 – 135	2 – 3
PVC – hart	170 – 210	20 – 60	30 – 50	70 – 200	12	0.2	70	1
PVC – weich	140 – 200	20 – 60	40	70 – 200	18	0.3	70	1
SAN	220 – 260	20 – 80	40	30 – 90	18	0.3	80	2 – 3
SB	180 – 280	20 – 80	20 – 30	50 – 100	30	0.6	80	1 – 2



5.6

Processing instructions for amorphous thermoplastics

* The data are guide values for dry air drying

5. Processing instructions

Partially crystalline thermoplastics

Material	Melt temperature [°C]	Mould temperature [°C]	Flange temperature [°C]	Specific back pressure [bar]	Peripheral speed [m/min]	Peripheral speed [m/s]	Drying temperature [°C] *	Drying time [h] *
PE-HD	180 – 300	10 – 60	20 – 30	15 – 45	60	1	**	
PE-LD	180 – 260	10 – 60	20 – 30	15 – 45	60	1	**	
PA 6	240 – 270	40 – 90	60 – 80	70 – 120	36	0.6	80	3 – 5
PA 6.6	270 – 320	40 – 90	60 – 80	50 – 150	36	0.6	80	3 – 5
PA 6.10	230 – 280	30 – 60	60 – 80	50 – 150	36	0.6	80	3 – 5
PA 11	200 – 250	20 – 100	60 – 80	50 – 150	36	0.6	85	4 – 6
PA 12	200 – 250	20 – 100	60 – 80	50 – 200	36	0.6	85	4 – 6
PEEK	350 – 380	150 – 180	100 – 120	2,0 – 50	12	0.2	150	2 – 3
LCP	295 – 385	80 – 120	60 – 80	0 – 30	12	0.2	150	4 – 6
LCP+GF	320 – 350	80 – 120	60 – 80	0 – 30	12	0.2	170	4 – 6
PBT	250 – 280	60 – 120	80	70 – 120	12	0.2	120	2 – 3
PC/PBT	255 – 270	40 – 80	40 – 60	50 – 120	24	0.4	110 – 120	2 – 4
POM	180 – 225	40 – 120	30 – 40	30 – 90	36	0.6	100	2 – 3
PP	200 – 260	10 – 60	20 – 30	150 – 450	54	0.9	**	
PPS	310 – 320	140 – 160	100 – 120	15 – 30	24	0.4	110 – 120	2 – 4
PPS+GF	330 – 350	140 – 160	100 – 120	15 – 30	24	0.4	130 – 150	2 – 4



5.7

Processing instructions for partially crystalline thermoplastics

*

The data are guide values for dry air drying

**

If these materials are filled with talc, they should be dried in accordance with the data provided by the manufacturers. (The data are guide values for dry air drying.)

5. Processing instructions

Partially to completely bio-based polymers

Material	Melt temperature [°C]	Mould temperature [°C]	Flange temperature [°C]	Specific back pressure [bar]	Peripheral speed [m/min]	Peripheral speed [m/s]	Drying temperature [°C] *	Drying time [h] *
PLA	170 - 210	25 - 60	20 - 50	20 - 60	60	1	50 - 60	4 - 6
PBS	140 - 230	10 - 40	not specified	80	30	0.5	70 - 80	4
PHA/PHB	160 - 175	50 - 65	not specified	5 - 15	12 - ≤ 24	0.2 - ≤ 0.4	80 - 90	2 - 4
CA	180 - 230	25 - 85	not specified	0 - 15	not specified	not specified	55 - 85	2.5 - 3.5
CP	190 - 235	40 - 80	not specified	0 - 3.5	not specified	not specified	55 - 85	1 - 3
CAB	180 - 235	25 - 85	not specified	not specified	not specified	not specified	55 - 85	2 - 4
PTT	250 - 270	80 - 110	80 - 90	0 - 3	12	< 0.2	120	2 - 4
PA 6.10	230 - 290	50 - 90	70 - 90	5 - 15	18	< 0.3	80 - 110	2 - 4
PA 10.10	215 - 290	60 - 100	40 - 60	50 - 100	6 - 18	0.1 - 0.3	80 - 100	2 - 6
PA 6.12	230 - 290	30 - 100	40 - 80	3.5 - 7	3 - 12	0.05 - 0.2	80 - 110	2 - 4
PA 11	210 - 270	30 - 90	not specified	0	not specified	not specified	80 - 90	4
PPA	300 - 330	110 - 180	40 - 80	5 - 15	6 - 18	0.1 - 0.3	100 - 120	4 - 6



5.8

Processing instructions for partially and completely bio-based polymers

*

The data are guide values for dry air drying

175 YEARS
KraussMaffei
Group



Quelle: Bayerisches Wirtschaftsarchiv

KraussMaffei Group

175 years of tradition in mechanical

175 years – Having reached this impressive anniversary, the KraussMaffei Group can look back on a spectacular success story within the international mechanical engineering industry. Founded in 1838, we have been manufacturing machines and equipment exclusively for the production and processing of plastics and rubber since 1999/2000. With a recognized premium level of product quality from the KraussMaffei, KraussMaffei Berstorff and Netstal brands, as well as high innovation capacity, the KraussMaffei Group is now a world leader in the plastics and rubber industries. We look forward to continuing to be your reliable and proven partner in the future.

Further information is available at:
www.kraussmaffei-group.com



Kunststoff-Institut

für die mittelständische Wirtschaft NRW GmbH

Karolinenstraße 8 | D-58507 Lüdenscheid

Tel.: +49 (0) 23 51.10 64 - 191 | Fax: +49 (0) 23 51.10 64 - 190

E-Mail: mail@kunststoff-institut.de | Internet: www.kunststoff-institut.de