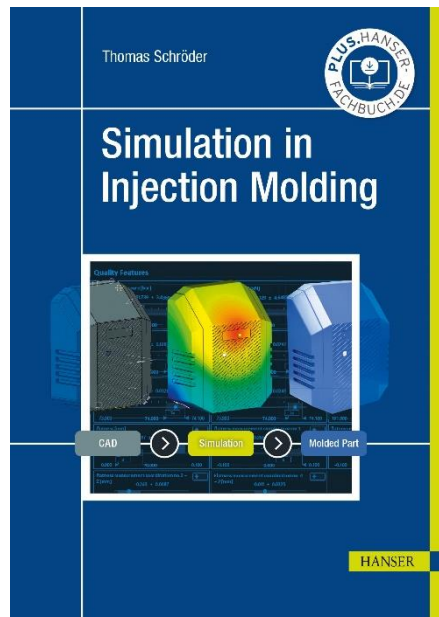


HANSER



Sample Pages

Simulation in Injection Molding

Thomas Schröder

Print-ISBN: 978-1-56990-916-4

E-Book-ISBN: 978-1-56990-932-4

For further information and order see

www.hanserpublications.com (in the Americas)

www.hanser-fachbuch.de (outside the Americas)

© Carl Hanser Verlag, München

Contents

Preface	XV
Foreword	XVII
The Author	XIX
1 Basics of Injection Molding	1
1.1 Components of an Injection Molding Machine	1
1.2 The Injection Molding Cycle	2
1.3 The Injection Mold	5
1.4 The Phases of Injection Molding – Forming the Molded Part	7
1.4.1 Injection Phase	8
1.4.2 Compression Phase	9
1.4.3 Holding Pressure Phase	10
1.4.4 Cooling Phase	11
1.5 Molded Part Faults	11
1.5.1 Gloss Differences	11
1.5.2 Knit Lines	12
1.5.3 Deformation	14
1.5.4 Warping	15
1.5.5 Diesel Effect (Burners)	16
1.5.6 Mold Coating (Deposits)	17
1.5.7 Black Specks	18
1.5.8 Ejector Marks	19
1.5.9 Burn Streaks	20
1.5.10 Sink Marks	22

1.5.11	Record Grooves (Cold Meld Lines)	23
1.5.12	Incomplete Filling of the Mold	24
1.5.13	Color Streaks	26
1.5.14	Glass Fiber Streaks	27
1.5.15	Moisture Streaks	28
1.5.16	Delamination	30
1.5.17	Tiger Stripes	31
1.5.18	Formation of Stress Cracks, Micro Cracks	32
1.5.19	Jetting	33
1.5.20	Voids	34
1.5.21	Ridges and Flash	35
1.5.22	Air Pockets	36
1.5.23	Cold Plug	38
1.5.24	Dull Spots near Gates or behind Cross-Section Transitions	39
1.5.25	Unusual Smell	40
1.6	Injection Molding Simulation	40
1.6.1	Simulation Calculation – Typical Methods	45
2	Rheological Basics	47
3	Introduction to Simulation Technology	53
3.1	Basics of Simplified Injection Molding Simulation	53
3.2	Basics of Extended Injection Molding Simulation	54
3.2.1	Uncertainties Regarding the Simulation Results	57
4	Mathematical Basics	59
4.1	Calculation Phase of Injection Molding Simulation	59
4.2	Calculation Variations	62
4.2.1	Finite-Difference Method (FDM)	62
4.2.2	Finite-Volume Method (FVM)	63
4.2.3	Finite-Element Method (FEM)	63

5	Types of Meshing, Errors in Meshing and Meshing Quality ...	65
5.1	Types of Finite Elements	65
5.1.1	Moldflow Meshing	67
5.1.1.1	Midplane (2D) Meshing	67
5.1.1.2	Dual Domain (2.5D) Meshing	68
5.1.1.3	Three-Dimensional (3D) Meshing	69
5.1.2	Difference Between 2.5D and 3D Calculation	70
5.1.3	Types of Moldex3D Meshing	71
5.1.4	Types of Cadmould Meshing	73
5.1.4.1	Cadmould 3D-F Meshing	73
5.1.4.2	Cadmould 3D-V Meshing	76
5.2	Impact of Meshing on Simulation Results	76
5.3	Possible Mesh Errors	80
5.3.1	Free Edges	80
5.3.2	Overlapping	81
5.3.3	Intersections	81
5.3.4	Aspect Ratio	82
5.4	Evaluation of the Meshing Quality	82
6	Material Cards in Simulation Programs	87
6.1	Setup and Process Data	88
6.2	Rheological Material Card	89
6.3	Thermal Material Card	92
6.4	Thermodynamic Material Properties – pVT Diagram	94
6.5	Mechanical Properties	98
6.6	Creating a Material Card	100
7	Easy Basic Exercises for Simulating Injection Processes	105
7.1	Comparison of Sprue Systems in Terms of Pressure Requirement	105
7.2	Flow-Path/Wall-Thickness Diagram	112
7.3	Calculating the Optimal Process Window during Injection	116
7.4	Sheet with Breakthrough – Filling Behavior	121
7.5	Sheet with Transverse Flow Stream toward the Rib	133
7.6	Sheet with Two Flow Directions and One-Sided Thin Spot	138

7.7	Parallel Flow Paths of Different Widths	143
7.8	Sheet with Central Thin Spot	147
7.9	Box with Thick Rim	151
8	Simulating the Molded-Part Formation Process in the Injection Mold	157
8.1	Injection Phase	158
8.1.1	Detection of the Optimal Injection Time and Speed	158
8.1.2	Comparison of the Real Injection Pressure with the Simulated Injection Pressure Based on a Flow Spiral	159
8.1.2.1	Flow-Path/Wall-Thickness Diagram – Comparing Reality and Simulation with the Standard Material Card	161
8.1.2.2	Flow-Path/Wall-Thickness Diagram – Comparing Reality and Simulation with the Customer Material Card	163
8.2	Holding Pressure Phase	164
8.3	Cooling Phase	167
9	Gating Systems in Injection Molding	169
9.1	General Information on Gating Systems	170
9.2	Cold Runner Systems for Thermoplastics	172
9.3	Hot Runner Systems	174
9.3.1	Simulation of Single-Point Gating with Open Hot Runner	178
9.3.2	Multi-Cavity Molds	183
9.3.2.1	Balancing Methods for Multi-Cavity Molds	186
9.3.2.2	Mathematical Rheological Balancing of Family Molds	197
9.3.2.3	Natural Rheological Balancing of a Multi-Cavity Mold	200
9.3.2.4	Thermal Balancing of a 2+2 Family Mold for Espresso Cups	211
9.3.3	Large Molds with Multi-Point Gating	216
9.3.3.1	Interpretation of Simulation Results for Standard Multi-Point Gating	219
9.3.3.2	Cascade Injection Molding	240
9.3.4	Influence of Sheared Edge Layers in Hot Runner Systems ...	257

9.3.5	Pressure Losses at Abrupt Changes in Cross-Section	259
9.3.6	Pressure Losses at Baffles	263
9.3.7	Simulation of Valve Gate System	266
10	Temperature Control Systems for Injection Molds	269
10.1	Example of Temperature Control	282
10.2	Factors Influencing Shrinkage and Warpage	292
10.2.1	Influence of the Temperature Control System on Shrinkage and Warpage	300
10.2.2	Influence of Fillers on Shrinkage and Warpage	302
10.2.3	Comparison of Simulated Shrinkage Values with Reality	309
10.2.4	Simulation of Shrinkage and Warpage Using Sample Molded Parts	311
10.2.4.1	Simulation of Shrinkage and Warpage with Pronounced Surface Temperature and with Real Cooling Circuit	311
10.2.4.2	Cooling Optimization Options	327
10.2.4.3	Variothermal Process Control	349
10.2.4.4	Comparison of Simulated Molded-Part Warpage with Measured Warpage Values	353
10.2.5	Void Formation during Injection Molding	373
11	Knit Lines and Meld Lines	377
11.1	Formation of Knit Lines and Meld Lines	378
11.2	Factors Influencing the Quality of the Knit Line	382
11.3	Effects of the Knit Line on the Mechanical Properties of the Molded Part	383
11.4	Factors Influencing the Knit Line Strength	386
11.4.1	Influence of Setting Parameters on Knit Line Strength	390
11.4.2	Influence of Venting on Knit Line Strength	392
11.4.3	Influence of Glass Fiber Content on Knit Line Strength	393
11.5	Simulation of Knit Line Strength	398
11.5.1	Comparison of the Simulated Strength of a Tensile Bar with Knit Line and without Knit Line	403
11.6	Examples of Knit Lines	404

12	Mechanical Design of Injection Molds	411
12.1	Mechanical Mold Design Based on Examples	417
12.1.1	Interior Frame Cover	417
12.1.1.1	The Injection Mold	418
12.1.1.2	Simulation Results	420
12.1.1.3	Mechanical Mold Analysis	422
12.1.1.4	Consequences of Mold Loading	423
12.1.2	Center Console	424
12.1.2.1	Center Console with Hot Runner	424
13	Simulation of Inserts	429
13.1	Simple Basic Example of a Simulation	429
13.2	Simulation of a Back-Molded Insert Made of Cast Iron	436
14	Core Offset Simulation	439
14.1	Examples of Core Offset Simulation	439
14.1.1	Push-Button	441
14.1.2	Automotive Fitting	451
14.1.3	Pump Housing	457
15	Integrative Simulation – Structural Analysis	463
15.1	Formation of Orientations during Injection Molding	466
15.2	Eulerian and Lagrangian Models	477
15.3	Material Models for Orientation Calculation	478
15.4	The Process of Integrative Structural Simulation	485
15.5	Simulation of the Injection Molding Process	486
15.6	Mapping with Digimat	496
15.7	Structural Simulation with Ansys Workbench	497
16	Sensor Technology for Injection Molding	507
16.1	Simulation of the Cavity Pressure Curve Using the Schäfer Box Mold	510
16.2	Simulation of Cavity Pressure and Specific Melt Pressure Based on the Flow Spiral	512
16.3	Process Control by Means of Sensors	516

17	Process Optimization by Means of Statistical Design of Experiments (DoE)	519
17.1	Robust Design of the Injection Molding Process	520
17.2	Use of Matrix Experiments to Characterize a Process	522
17.3	Distinction between “Robust Design” and Statistical Design of Experiments	526
17.4	Process Optimization Using Autodesk Moldflow	532
17.5	Process Optimization Using Cadmould Varimos	540
17.6	Selection of the Quality Characteristics	543
17.7	Analysis of the Statistical Experimental Design/Optimization	547
17.8	Weighting and Reduction in Quality Characteristics	552
18	Special Injection Molding Processes	555
18.1	Sandwich Injection Molding Basics	555
18.2	Simulation of Sandwich Injection Molding	559
18.3	Fundamentals of Gas Injection Molding	564
18.3.1	Simulation of Gas Injection Molding: Chainsaw Handle	565
18.3.2	Simulation of Gas Injection Molding: Air-Filter Housing	567
18.3.3	Simulation of Gas Injection Molding: Large-Area Cover	571
18.3.4	Simulation of Gas Injection Molding: Large-Area, Thick-Walled Molded Part	575
19	Crosslinking Molding Compounds	579
19.1	General Information on Crosslinking Molding Compounds	579
19.2	Flow Behavior of Crosslinking Molding Compounds	580
19.3	Simulation Example for Liquid Silicone Rubber (LSR)	581
19.3.1	Mold Technology for LSR Molding Compounds	582
19.3.2	Curing Kinetics of LSR Molding Compounds	583
19.3.3	Optimization of the Gating Position of the Potholder	585
19.3.4	Mold Optimization of the Potholder with Regard to Curing Time	589
19.3.5	Venting the Potholder Mold	590
19.3.6	Optimization of the Heating System of the Potholder Mold ...	591
19.3.7	Simulation of Thermal Buildup in the Potholder Mold	596

20	Mesh-Friendly Optimization of CAD Models with CATIA V5	599
20.1	Evaluation of the Input Data	601
20.2	Guidelines for Optimizing the Data	603
20.3	Analysis of the Input Data	607
20.3.1	Assembly or Single Part	608
20.3.2	Closed Volume in a Composite Body in the Single Part	612
20.3.3	Multiple Bodies in the Single Part	613
20.3.4	Polysurface with Individual Gaps in the Single Part	618
20.3.5	Positioning of Several Parts Does Not Fit Together	619
20.4	Dress-Up of a Polysurface with Gaps	623
20.4.1	Tools of Generative Shape Design	624
20.4.2	Notes on the Organizational Structure in Generative Shape Design	625
20.4.3	Joining Existing Surfaces	628
20.4.4	Disassembling the Polysurface into Domains	629
20.4.5	Localizing Gaps (Boundaries)	629
20.4.6	Removing Surface Segments (Sub-Elements)	630
20.4.7	Enlarging Existing Surfaces	631
20.4.8	Splitting Open Surfaces at Boundary Curves	633
20.4.9	Splitting a Closed Polysurface at Boundary Curves	634
20.4.10	General Information on Generating Shapes with the Help of Neighboring Geometry	635
20.4.11	Generating a Blend	636
20.4.12	Generating Fill	638
20.4.13	Creating a Closed Polysurface	638
20.4.14	Closing the Polysurface to Volume	640
20.5	Simplifying the Geometry to Reduce the Calculation Effort	641
20.5.1	Exploiting Symmetries	642
20.5.2	Convenient Removal of Radii	642
20.5.3	Manually Removing Radii	647
20.5.4	Removing Lettering	652
20.5.5	Removing Design Elements	662

20.6	File Output	670
20.6.1	stp Format	670
20.6.2	stl Format	673
20.7	Removing the Optimization History	673
20.8	Joining Bodies Using Boolean Operators	676
20.9	Collision Analysis as a Means of Control	676
20.10	Intersection of Areas as a Means of Control	677
20.11	Coupling Hot Runner Nozzles to the Cavity	677
20.12	Positioning and Scaling of Core and Molded Part	680
Index	683

Preface

Simulation software programs are now a staple in the field of injection molding for the design of molded parts and molds. They assist designers early in the development of the molded part and mold design, thereby reducing the often time-consuming modification cycles. This shortens development time and lowers development costs. Additionally, the programs can enhance the quality of the molded parts and the efficiency of the injection molding process. Given the wide range of simulations that these programs can perform, the breadth of the simulation results is equally extensive. The challenge usually lies not in generating the results produced, as these programs are all very user-friendly, but in understanding and interpreting them. Only when this is the case can the user draw meaningful conclusions from them and suggest potential improvements. This is exactly where this book comes in. It begins with a general introduction to the injection molding process, meshing, and basic mathematics, followed by simple basic exercises. These are designed to provide the reader with an initial understanding of the simulation technique of how to interpret the results. Each chapter then explains the prerequisites for injection molding, the framework conditions and mathematical correlations, before conducting sample simulations out on real parts. For each specific simulation case, the results are interpreted and solutions developed. Finally, the aim is to use these simulations to make the reader aware of the essential points, which can vary extensively from part to part.

As the book contains numerous pictures and graphics and these are not always very easy to see due to their size, we have decided to provide the reader with all the pictures additionally in the form of image data. At *plus.hanser-fachbuch.de*, you can access the images using the code on the first page of the book.

Darmstadt 2024

Dr. Thomas Schröder

Foreword

For the most successful companies in plastic injection molding, simulating every part in advance has become the standard way of working. Whether you're a seasoned professional or still learning the ropes, simulation amplifies your skills, allowing you to identify and prevent process or part issues before any metal is machined. This includes issues like filling errors, warpage or excessive cycle times, among others. Another key advantage that is often overlooked is how simulation results and images bring clarity and focus to discussions with colleagues, superiors, and customers. By visualizing potential outcomes you can have a more constructive discussion, because your customers and colleagues will more quickly understand what you mean.



Naturally, mastering such a powerful tool takes some practice. You need to gain a deep understanding of how to represent reality as accurately and efficiently as possible if you want accurate results. But even the best simulation results are only as valuable as the insights you draw from them. Simulation is not just a prediction tool – it's a decision-making tool. That's why Prof. Dr. Thomas Schröder's focus on correctly interpreting simulation results is spot-on. His 30 years of expertise in plastics technology, distilled into this book, will save you from many potential missteps, and help you capture the full value of simulation.

Mastering the insights from this book can significantly advance your career. Given the pace at which engineering departments are digitalizing their ways of working, skilled simulation experts are increasingly in demand. By reading this book, you will not only enhance your professional knowledge but also your market value.

Ines Oud

CEO of injection molding simulation company **SIMCON** and Chairwoman of the German plastics industry association **Kunststoffland NRW**

The Author

Prof. Dr. Thomas Schröder teaches injection molding, rheology, mold technology and simulation technology at Darmstadt University of Applied Sciences (h_da Hochschule für Angewandte Wissenschaften), Germany, and is a member of EUt+ (European University of Technology). After studying mechanical engineering with a specialization in plastics technology at RWTH Aachen University, he completed his doctorate under Prof. Dr. Dr. h. c. Walter Michaeli on the subject of gas injection technology. After working for several years at a well-known plastics manufacturer, he moved to Krupp Corpoplast in Hamburg, where he was responsible for injection molding systems that produce preforms. Following this position, he moved to Netstal Maschinen AG in Näfels, Switzerland. There, he headed the application technology SPA of the injection molding machine manufacturer until he received a call to the University of Applied Sciences Darmstadt in 2001. He is also the author of the book *Rheologie der Kunststoffe* (Rheology of Plastics), the second edition of which was published by Hanser Verlag in 2020. Prof. Dr. Thomas Schröder is a member of the Institut für Kunststofftechnik Darmstadt ikd and chairman of the Gesellschaft zur Förderung technischen Nachwuchses GFTN e. V. (Society for the Promotion of Young Technical Talent) at Darmstadt University of Applied Sciences. Furthermore, he is a member of the PhD Centre for Sustainability Sciences and has the right to award doctorates, due to his numerous research projects, i. e., third-party funded projects in the field of injection molding, rheology, simulation, and tool technology. Prof. Dr. Thomas Schröder is also the managing partner of PlastSolutions Consulting GmbH, based in Mannheim.



Acknowledgments

This book could not have been written without the invaluable cooperation of many Bachelor's and Master's students. In this respect, I would first of all like to thank the many graduates whom I was able to supervise in various subject areas. I wish you all the best and much success in your professional career and also privately! In

particular, I would like to thank the following people for their active participation in the book project: Aletta Berger, Sophie Dolata, Mary Göhler, Laura Gollan, Markus Hammermeister, Fabian Nebel, Bardo Palmberg, Bastian Paulsen and Maximilian Teich.

Furthermore, I would like to thank the company Simcon, which has always supported me with the software modules Cadmould and Varimos. Finally, I would like to thank the company Mold and Hotrunner Technology MHT, which also assisted me with the preparation of this book.

1

Basics of Injection Molding

■ 1.1 Components of an Injection Molding Machine

Almost one third of all plastics are processed by injection molding. This master-molding technique is a discontinuous process. Injection mold machines are used for producing moldings from thermoplastics, elastomers and thermosetting plastics. These machines consist of several modules such as the machine bed, the plasticizing and injection unit, the clamping and locking unit, the temperature control unit as well as the mold and the control unit (Figure 1.1) [1].

The machine bed serves to accommodate the clamping, locking and injection units. In addition, pumps and tanks for hydraulics oil are built into the base of machines that need hydraulics. In fully electric injection molding machines, electric motors drive the five main movement axes (dosage, injection, plasticizing and injection unit, machine movement and ejection). The control cabinets are on one hand necessary for accommodating the mold and on the other hand for raising the clamping forces. These forces prevent the mold from opening during the shaping process. Another one of their tasks is closing and opening the mold. The mold is responsible for the shaping of the plastics and releasing the warm air so that a molded part can be formed out of plastics melt. The plasticizing and injection unit is used to melt and supply the plastic. The plastic granules are added through a hopper into a screw that is surrounded by a heated cylinder. The movement of the screw and the thermal conduction of the cylinder plasticizes the granules into a melt, which is pushed into the antechamber of the screw. The melt is then pushed forward into the closed mold. The control unit is needed for adjusting and observing the setting parameters whereas the temperature control unit regulates the temperature of the mold at a pre-set temperature [1].

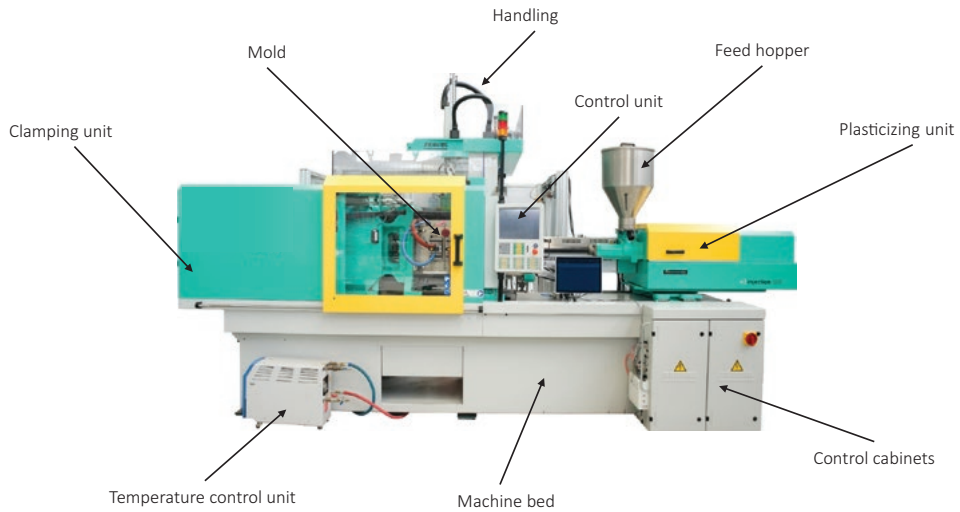


Figure 1.1 Set-up of an injection molding machine

Injection molding machines can be differentiated on the basis of their design and drive type. Thus, a distinction can be made between horizontal and vertical injection machines and fully electric, hydraulic and hybrid machines. The difference between a horizontal and vertical injection machine is the position of the mold or the molded parting line in the machine. The type of drive used determines whether the injection machine is fully electric, hybrid or hydraulic. The axes of fully electric machines are driven by an electric motor. Hydraulic machines are driven by hydraulics while hybrid machines have a mixture of hydraulics and electrically driven axes [2].

■ 1.2 The Injection Molding Cycle

A typical injection molding cycle runs in five steps. A new cycle commences when the mold closes. As soon as the mold is closed and the clamping forces are built up, the plasticizing and injection unit is pushed forward onto the mold. Then, the molten plastic (plastic melt) is injected into the cavity under a speed-controlled axial advance of the screw. The holding pressure phase commences when the mold is 95–98% filled and serves to regulate shrinkage of the molded part. In this phase, the holding pressure is regulated and the plastic melt is pushed further into the mold to balance the volume shrinkage caused by the temperature. The phase ends when the connection of the sprue to the molded part solidifies. This means that the sealing point has been reached. From this point on, the plasticizing and injection

unit of the mold returns to its original position and the so-called residual cooling time begins. During this phase, the dosage process for the next cycle begins, which means that plastic melt again fills up the antechamber of the screw. As soon as the molded part reaches its ejection temperature, the mold opens and ejects the molded part. Alternatively, a handling system removes the molded part from the mold. Figure 1.2 shows the injection molding process.

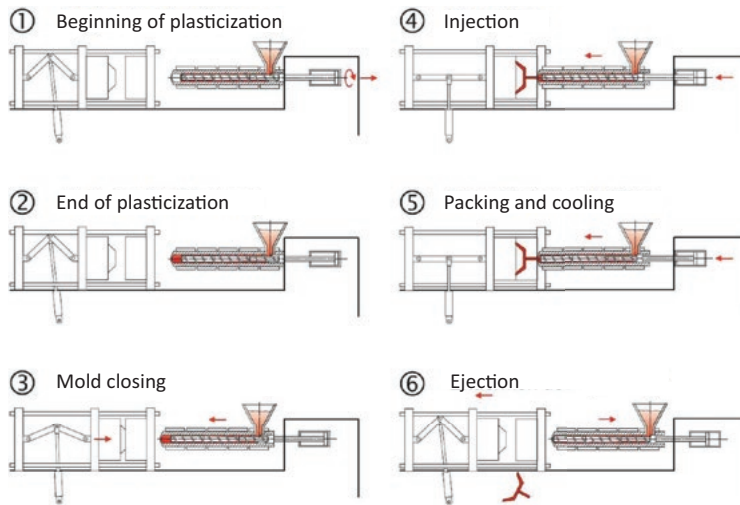


Figure 1.2 The injection molding cycle [1]

The cycle time consists of the time needed for opening and closing the mold, the deployment of handling and ejectors, as well as the injection time, the holding pressure time and the residual cooling time. It is essential for the financial planning to give consideration to the cycle time, as reducing it is one of the most important aims of the injection molding process. Based on the cycle time, the mold and number of cavities can be calculated for the required number of molded parts per year. Figure 1.3 shows a typical injection molding cycle [2, 1].

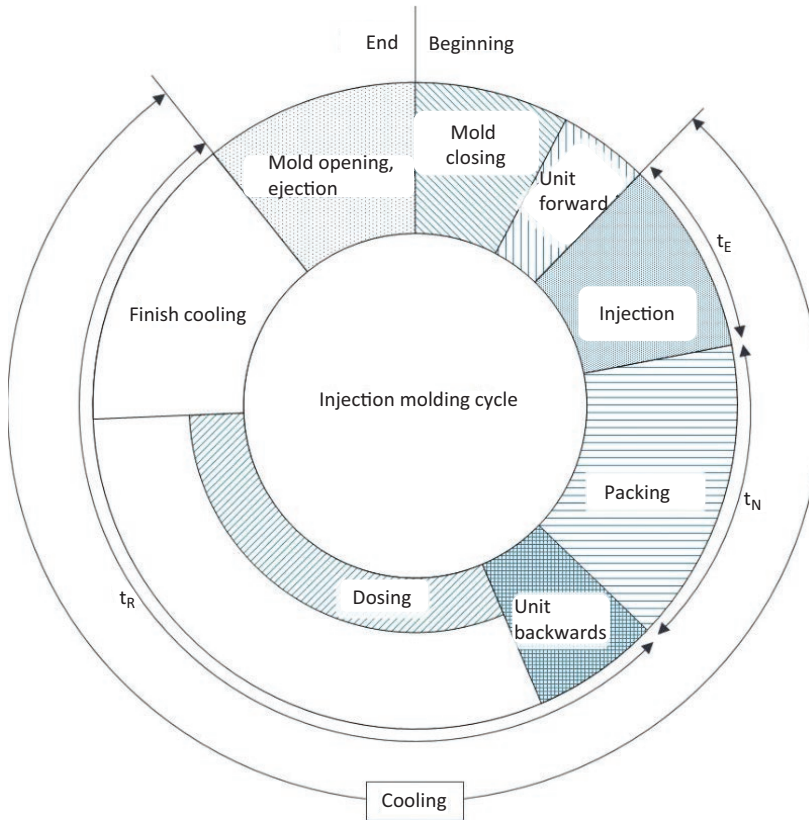


Figure 1.3 Injection molding cycle

The run time of the cycle can be calculated as follows (Equation 1.1):

$$t_z = t_E + t_N + t_{RK} + t_{WZopen} + t_{Ejector/Handling} + t_{WZclose} \quad (1.1)$$

Where:

t_z = Cycle time

t_E = Injection time

t_N = Holding pressure time

t_{RK} = Residual cooling time

$t_{Ejector/Handling}$ = Ejection time

$t_{WZopen/close}$ = Mold opening and closing

As soon as the mold is closed, the injection phase begins. This phase also affects the cooling time. The reason for this is that, when the material enters the mold, the plastic melt touches the wall of the mold and that starts the cooling process. The holding pressure phase is needed to balance the volume shrinkage when the molded part solidifies. During this phase, a specified holding pressure pushes more plastic melt into the mold.

Immediately after the holding pressure phase is concluded, the residual cooling time before the molded part is ready to be ejected begins. During the residual cooling time, the granules for the next shot are being prepared, that is to say, plasticized. Once the molded part is stable enough to be ejected, the mold opens and the part can be pushed out of the mold by the ejectors. It is also possible to take out the part by means of a handling system. The time required for this needs to be taken into consideration when calculating the cycle time.

■ 1.3 The Injection Mold

Generally, each injection mold consists of two mold halves (Figure 1.4): one is fixed (nozzle side) and the other is movable (ejector side). The molded part is formed by the cavity. The plastic melt streams through the sprue bush until reaches the cavity. The sprue bush is located on the fixed-nozzle side and, during the injection phase, it is in contact with the plasticizing and injection unit. In general, the cavity consists of a cavity plate, which usually is on the nozzle side, and a core that sits on the ejector side.

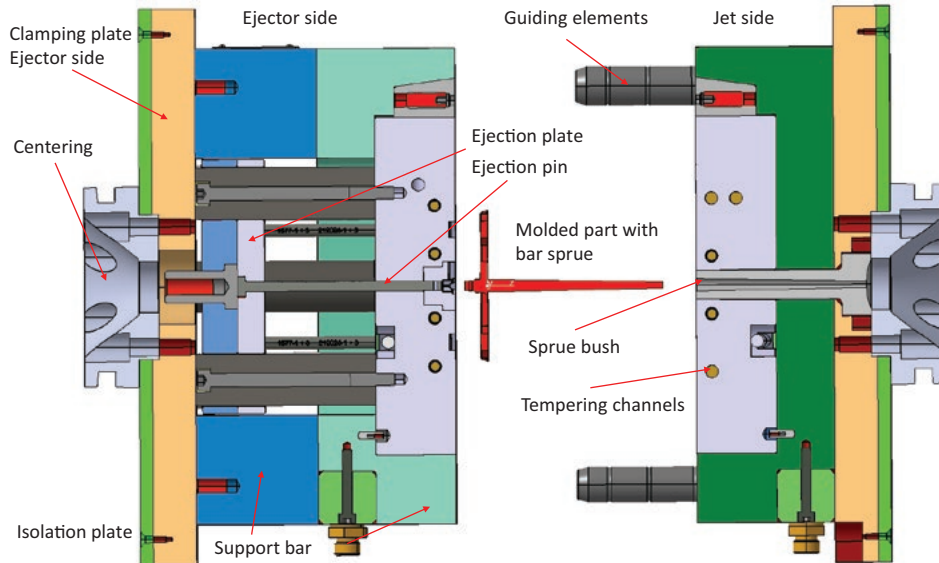


Figure 1.4 Structure of a two-plate injection mold (Source: PlastSolutions GmbH)

Cooling channels in both mold halves are necessary to dissipate the heat of the molded part. A cooling medium (usually water) flows through these cooling channels. With water and under pressure, cooling is possible up to 200 °C. For higher temperatures, oil is used in most cases to dissipate the heat. However, water has better heat transfer properties than oil. The goal is to achieve turbulent flow, as that allows for better heat transfer. The cavity plate as well as the core are located in the mold plate. After the molding process, when the molded part has reached the ejection temperature, the ejector pins remove the molded part from the core on the ejector side. During this process, an ejector bolt is pushed forward, usually by a hydraulic ejector. The ejector plate in which the ejector pins are located push the molded part out of the ejector side.

The supporting bars are behind the intermediate plate and in between the supporting bars is the ejector plate. For fixing the mold on the movable machine plate, a clamping plate is located at the end of the ejector side. The same applies to the nozzle side. Here, the clamping plate is located at the end of the fixed plate to attach this side to the fixed machine plate. If necessary, an additional isolation plate can be added to these to plates to reduce the heat transfer from the mold to the clamping plate. It is possible to install a centering ring on the nozzle side as well as the ejector side. This aids the installation of the mold inside the machine and the centering of the mold (center line to the plasticizing unit). Guide elements guide the mold and hold it together when not assembled.

One of the main tasks of the injection mold is to let the plastic melt stream from the plasticizing and injection unit through the sprue bush and finally into the cavity. In the cavity, the heat of the plastic melt needs to be reduced so that the molded part can be cooled down to ejection temperature. This means that the mold is tasked with giving the molded part its form and transferring the heat. After the ejection temperature is reached, the ejectors push out the molded part. In general, it is important that the mold is rigid enough to withstand the high pressure during the injection phase as well as deformation due to the high clamping force.

Injection molding simulation programs can interpret injection molds rheologically, thermally, and mechanical. This means that it is possible to simulate the flow of the plastic melt from the tip of the screw, to the sprue system to the end of the flow path. Finally, the injection molding simulation program is able to simulate the holding pressure and cooling phases. This allows the pressures, speed, shear rate and stress, temperatures and other measurements to be depicted during the injection, holding pressure and cooling phases. The injection molding simulation process and the cooling can be optimized and the shrinkage and warpage can be predicted. A prediction of the shrinkage and warpage enables early optimization of the injection mold and, if necessary, adjustment of the mold [3]. Statistical methods (Design of Experiments or DoE) support the user of the simulation program during the evaluation of the major parameters on the qualitative features and the process optimization [4].

Current simulation programs make it possible to simulate the starting process of an injection molding machine. As it takes some time for the thermal balance of the injection mold to be reached, simulation of the startup process is especially important. In addition to the deformation of individual mold elements, it is also possible to predict the offset of the core. A mechanical interpretation of the mold can also be executed by the simulation programs.

■ 1.4 The Phases of Injection Molding – Forming the Molded Part

Generally, the molded part is produced in three phases:

- injection phase (dynamic phase)
- compression phase (quasi-static phase)
- holding pressure phase (quasi-static phase)

These three phases define the quality of the plastic molded part. Figure 1.5 shows the cavity pressure gradients and the three phases of molded part formation during injection molding.

All the main quality features, such as weight, dimensional stability, surface quality and so on are mostly formed during these three phases. Therefore, it is crucial to know the pressure gradients during these phases. Computer-aided simulation programs can be of service here as well.

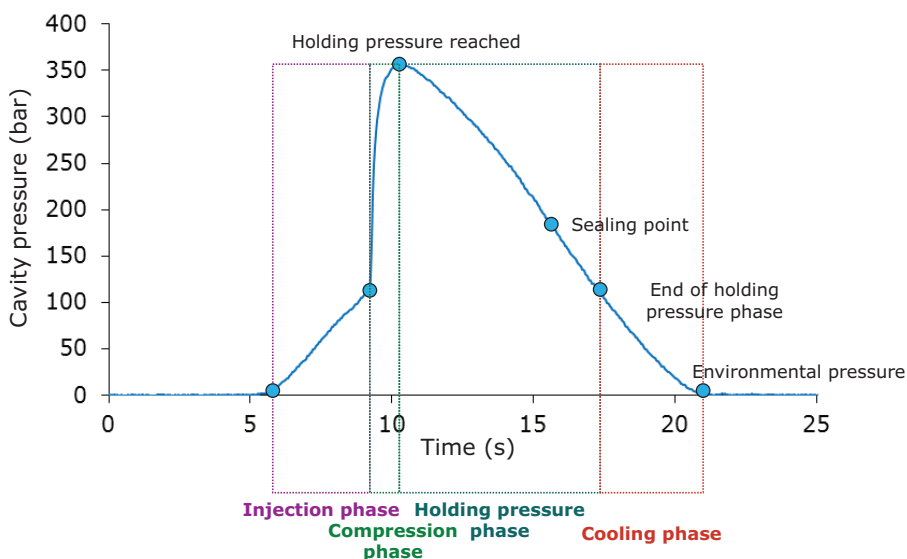


Figure 1.5 Cavity pressure curve during injection

These programs help with making statements about the formation of the molded part (pressure, flow rate, shear, shear stress, temperature, etc.). As the process of forming the molded part (Figure 1.5) is directly connected to the quality of the molded part, it is essential to be aware of possible weak spots or faults in the molded part in advance. A high level of expert knowledge is generally necessary since the correlations in this process are usually very complex.

1.4.1 Injection Phase

In the dynamic injection phase, the liquid plastic that has been prepared by the plasticizing and injection unit is injected into the closed mold by the axial advance of the screw. This process is usually speed-controlled. This means that the injection molding machine provides the injection pressure (up to 2600 bar) necessary for the filling of the mold. Therefore, it is crucial that the switching point, where the speed-controlled injection phase switches over to the pressure-controlled holding pressure phase, is precisely timed. The switching point has to occur somewhere between 95 and 98% of volumetric filling of the cavity. The injection phase is followed by the compression phase.

In the injection phase, the surface quality of the part is defined by fountain flow at the melt front and the wall adhesion of the plastic melt. Furthermore, the orientations, caused by the flow direction, of molecular chains and fillers, such as glass fiber, are formed. If the switching point takes place at the right time, the cavity pressure will look similar to Figure 1.6.

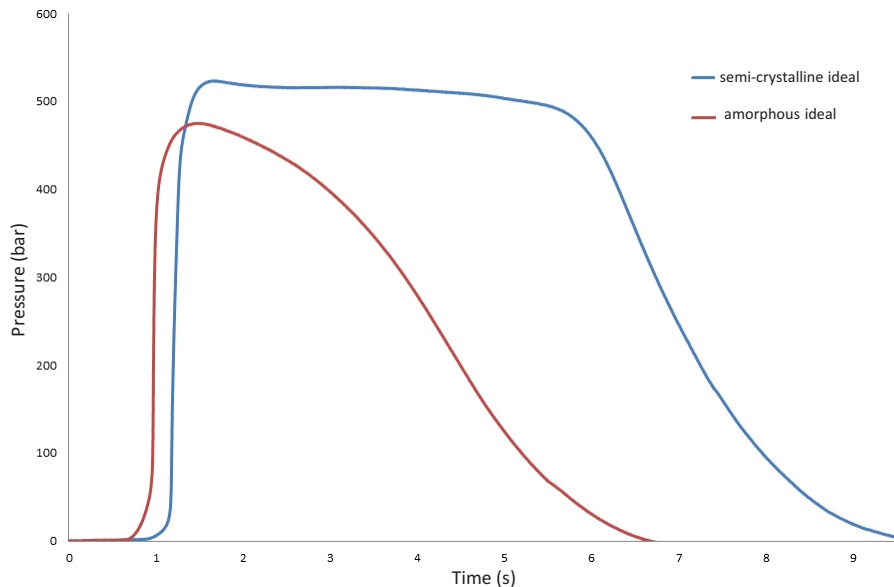


Figure 1.6 Cavity pressure at the right switching point during injection molding

Figure 1.6 shows the cavity pressure for an amorphous polymer (red) and a semi-crystalline polymer (blue). The curve for the amorphous polymer tends to be harmoniously round, due to the cooling behavior. On the other hand, the curve for the semi-crystalline polymer shows a strong holding pressure level at first. Next, the cavity pressure decreases rapidly. The standard switching point in simulation programs is usually set at 98–99% volumetric filling. This value does not necessarily have to be the correct point for the switching process. The same applies to the injection time or speed. The plausibility of these standard values should always be checked in a simulation.

1.4.2 Compression Phase

The quasi-static compression phase follows the injection phase. This phase heavily depends on the switching point. If the switch from injection to compression happens too early, the cavity pressure will collapse (Figure 1.7, left). Also, the volumetric residual filling of the cavity is pressure-controlled and this affects the quality of the molded part (surface, orientations, etc). Even if the switch has only been executed after 99% of volumetric filling, the injection molding machine will try to keep the injection speed in a constant balance, even if the cavity is already completely filled. This leads to high pressure peaks inside the mold (Figure 1.7, right).

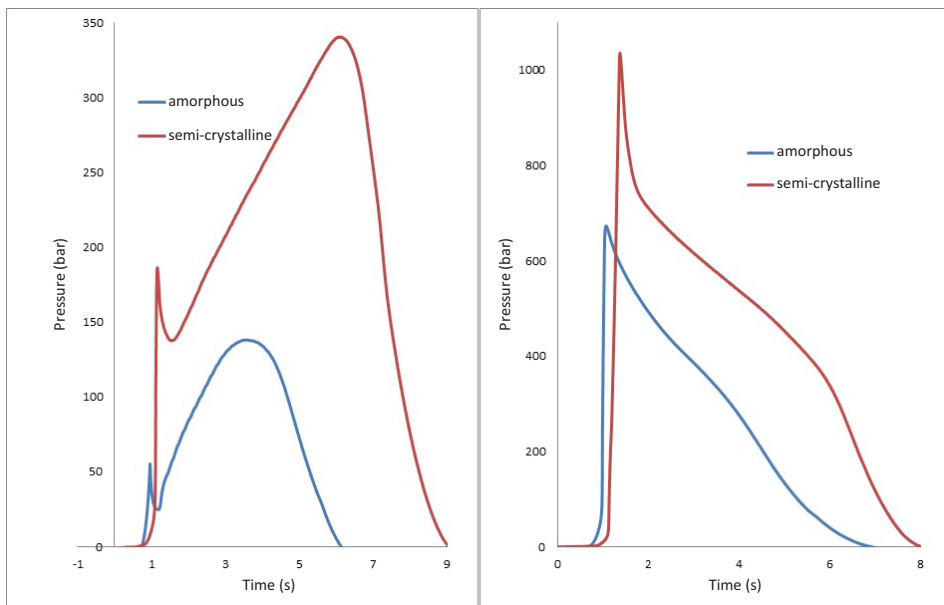


Figure 1.7 Cavity pressure at the wrong switching point during injection molding

In some circumstances, the clamping force exerted by the injection molding machine may not be sufficient if the switching point is set too late. As a result, the mold may open. This can cause overfilling and, directly related to this, the formation of burrs. During the cooling phase, the burr remains in the mold parting line. In this phase, the molded part peels away from the cavity surface. During this process, the clamping force of the machine puts pressure on the burr (web) that is situated in the molded parting line. As a result, damage to the mold may occur. Additionally, the high cavity pressure may cause the plastic melt to be pushed back into the plasticizing unit. This is possible when the cavity pressure is higher than the set holding pressure and when the machine or the mold has no needle gate nozzles.

1.4.3 Holding Pressure Phase

The quasi-static holding pressure phase is supposed to compensate (balance) for volume shrinkage of the molded part. The volume shrinkage, which is caused by the cooling of the molded part and the related relaxation processes of the molecules, is compensated by the further feeding of hot plastic melt under high pressure. The user of an injection machine usually chooses a percentage of the pressure (the melt pressure in the ante-chamber) as the starting point for the holding pressure. A starting point could be 30% of the injection pressure. This starting value can be increased until the desired part quality (dimensional accuracy, sink marks, voids, weight, etc.) is reached. The holding pressure should only be set as high as necessary, as a high holding pressure can impair part quality (residual stresses, orientations, etc.). Additionally, a high holding pressure puts strain on the machine and the mold, such that energy usage increases.

The simulation programs usually have a guide value, mostly listed in tables, for the level of the holding pressure. This value depends on the type of plastic and the part geometry. The simulation programs do not know the injection pressure, more specifically the melt pressure in the ante-chamber. Therefore, they need to use the values in the tables for their calculations. It is therefore necessary to check the values and compare them to practical experience. The same considerations also apply to the holding pressure and the cooling time.

The holding pressure time ends after the so-called sealing point has been reached. Usually, the sprue that has a thin wall freezes first and causes the on-going holding pressure to stop. The curve for the cavity pressure shows the sealing point of an amorphous polymer very clearly (Figure 1.7, red) at the turning point of the decreasing holding pressure.

The simulation programs use these curves not to calculate the sealing point, but to define a flow limit or no-flow temperature. This no-flow temperature is calculated from the p v T data and the specific heat capacity. This is the limit value which determines when the plastic freezes and stops melting.

1.4.4 Cooling Phase

The plastic part is cooled down to ejection temperature in the cooling phase. In the literature, a distinction is drawn between a medium and a maximum ejection temperature. While the cooling equation for the medium ejection temperature is more likely to be used for amorphous polymers, the equation for the maximum ejection temperature is preferably used for semi-crystalline polymers. A precise value is set in the simulation programs. The ejector temperature can be calculated from the curve of the 1 bar line in the pVT diagram. For amorphous polymers, the ejector temperature is 30 to 50 °C below the glass temperature (kink in the 1 bar line). For semi-crystalline polymers, a tangent is drawn to the curve of the transition area from fluid to solid. The point of intersection is the ejection temperature.

■ 1.5 Molded Part Faults

Several different faults can occur on the plastic molded part during the injection molding process as a result of a variety of influential parameters (settings, machine, mold, material, etc). As a wide range of aspects could be the cause of these faults, it is often difficult to identify them and in turn to solve the issue. Many faults or flaws can be identified in advance by the simulation. Consequently, appropriate measures can be applied in the simulation to rectify the issues. Therefore, executing the simulation early can help to reduce costs significantly and avoid time-consuming corrections on the mold later on.

1.5.1 Gloss Differences

During the injection molding process, differences in the gloss of the molded part surface can appear. There are two reasons for their occurrence: either more or less severely glossy spots are formed on the part surface despite a uniformly structured cavity, or the gloss of the whole surface area is too intense or weak. In general, the intensity of the gloss is impacted by molding of the surface cavity (Figure 1.8). It is possible to achieve a more matt surface when the molding has been done well. In addition, the surface of the mold must have some structuring, as these bumps result in a diffuse, non-targeted reflection of light. However, if the surface of the mold is polished, a well-executed molding process results in a more intense glossy surface of the molded part. The mold and melt temperature are critical to these processes, as well as the injection temperature and the holding pressure. The settings significantly influence the development of the frozen edge and therefore also

the molding of the surface. Fundamentally, it needs to be noted that the surface quality of a plastic molded part is primarily defined by the flow of the melt in the injection phase.

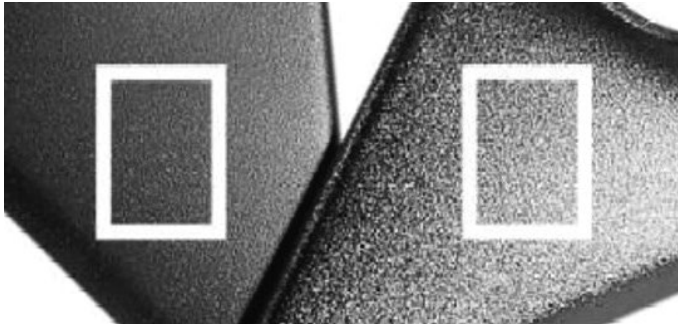


Figure 1.8
Gloss differences

Possible solutions:

- Optimize the injection speed (usually: increase the speed).
- Increase the mold temperature (the temperature limit has to be observed). However, a higher mold temperature extends the cooling time and the cycle time.
- Increase the melt temperature (the temperature limit has to be observed). The melt temperature also influences the length of the cooling time but not as much as the mold temperature.
- Increase the holding pressure and, if necessary, extend the holding pressure time.
- Delay the switching point to holding pressure until the fill level has reached 98% to achieve a process optimization.

1.5.2 Knit Lines

Knit lines (weld lines) cannot be avoided during injection molding (Figure 1.9). They can develop behind a breakthrough, through several flow paths, as a result of inserts, through differences in wall thickness, or due to several gating points. Knit lines develop when two melt flow fronts collide more or less head-on. Notches can appear on the surface of the plastic part near the knit lines. In most cases, differences in gloss and color are also visible. Knit lines can be notably spotted on transparent or dark plastic parts with a polished surface or plastics which contain effect pigments or glass fibers. If the melt meld lines meet under overly low pressures and temperatures, notches form on the surface in this area. The melt meld lines then do not properly weld together. Due to this reason, mechanical strength is most

likely reduced under these circumstances. Another problem that can occur with knit lines is the orientation of the reinforcing fillers such as glass fibers. The smaller the angle between the melt flow fronts, the more extreme the distinctness of the knit line usually is.

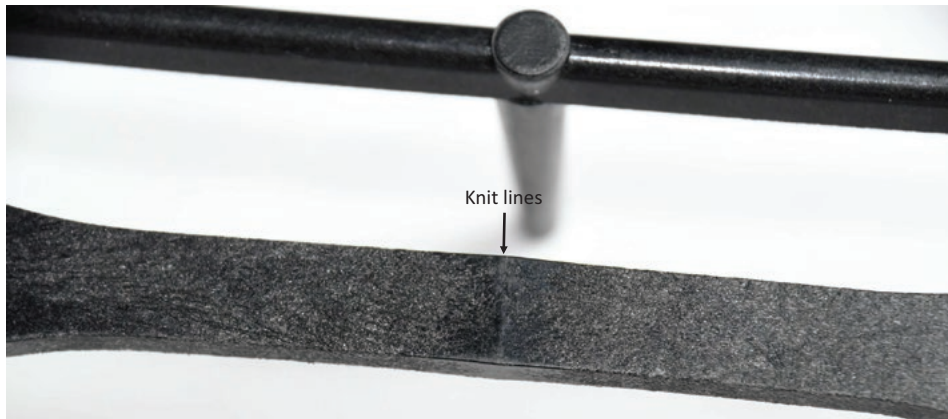


Figure 1.9 Knit lines

Possible solutions:

- Optimize the injection speed. If the venting is good enough, the injection speed should be increased.
- Use a filling study of the plastic molded part to check the venting. If the air cannot escape properly near the knit line, it will be compressed. This may result in the diesel effect. In this case, it is necessary to implement venting in this area. Venting can be achieved by the implementation of more ejectors or special inserts, such as disk packs, as well as a splitting of the mold insert into diverse small parts.
- Increase the mold temperature (the temperature limit has to be observed). However, a higher mold temperature extends the cooling time and the cycle time. It is also possible to regulate the temperature of local inserts in the mold (variothermal dynamic temperature; see Chapter 10). This way, it is possible to heat the knit lines in the injection phase locally. As a result, the quality might increase or the knit lines may even become invisible.
- Increase the melt temperature (the temperature limit and elongation of the cycle time have to be considered).
- Delay the switching point to holding pressure until the fill level has reached approximately 98%; this could lead to process optimization.
- Increase the holding pressure to improve the welding of the two melt flow fronts.

- (Reinforcing) fillers and pigments usually impair the quality of the knit lines.
- Move the gating point to influence the position of the knit lines due to the wall thickness. If necessary, the wall thickness can be adjusted. However, retroactively this is only possible by modifying the mold. Therefore, the effectiveness of this measure should be checked in advance with a filling simulation. The simulation programs show the knit lines in color as a function of the angle of incidence. Consequently, adjustments to the geometry of the plastic part, such as the gating point, are easy to make.

1.5.3 Deformation

Deformation can occur when overly strong or irregular forces impact the ejection of the plastic part (Figure 1.10). Usually, this can be seen in the material, which expands, tears, or even breaks due to the high ejection forces. In most cases, in contrast to warping, deformation occurs near the area of the ejector or on undercuts that are difficult to demold. Moreover, deformation can cause deep mold marks or scratches in the direction of demolding.



Figure 1.10
Deformation

Possible solutions:

- Check the draft angles. After they have been evaluated, the drafts should be increased if necessary.
- Check the position and the number of ejectors.
- To avoid deformation, adjust the holding pressure.
- Add a lubricant or anti-adhesion coating to the cavity.
- Reduce the temperature of the mold core.
- Optimize the core venting. This can be achieved by installing more venting, special inserts, or ejectors, etc., through which the air can flow between the molded part and the core.

1.5.4 Warping

Differences in volume shrinkage in different areas of the molded part can cause deviations from the planned shape. These deviations are called warping (Figure 1.11). It is visible through warps, wave-like surfaces, torsion or deviations in angles. The difference between holding pressure forces (shrinkage compensation) and orientations cause the differences in shrinkages. Another cause of warping could be internal stress.



Figure 1.11
Warping

Possible solutions:

- Cool the molded part evenly. This means that the temperature of the mold needs to be checked.
- Ensure sufficient compensation of volume shrinkage in the holding pressure phase. To achieve this, choose a holding pressure that is sufficiently strong and long.
- To avoid warping, increase the injection speed.
- Uniform filling process of the mold is advisable.
- Choosing a material that flows more easily can be beneficial.
- Another feature should be considered when choosing the material: low shrinkage values. Semi-crystalline and unfilled plastics tend to shrink more than plastics with fillers and amorphous polymers.
- Several sprues improve the holding pressure forces and therefore reduce warping.
- Uniformly distributing the melt along the flow path could lead to less warping.
- Avoid sharp corners.

- Reinforce areas where warping primarily occurs (ribs).
- Check whether the fiber orientation can be changed.
- Prevention of warping should already be considered when manufacturing the mold by ensuring that the molded part with warping corresponds with the reference part.
- Reduce differences in wall thickness and unusual accumulations of melt.
- Warping can be minimized if convex areas or edge beads in which the deformation has no impact can be formed. Simulation programs can predict warping and suggest preliminary corrections in the mold, Cadmould with Unwarp, for example. However, these suggestions need to be critically examined before implementation. For this reason, Cadmould issues a warning (see Chapter 10).

1.5.5 Diesel Effect (Burners)

If the air cannot escape properly during the injection phase from the mold, black discoloration may appear near the knit lines, where the melt flows collide, or in specific areas, such as the ribs or at the end of the flow path. In this case, the air gets compressed during the filling process, so that the diesel effect occurs (Figure 1.12). In isolated cases, it can also prevent the molded part from filling completely. This could also result in a layer forming on the surface of the mold or even thermal damage. The compressed air gets heated so much that burned spots or burners can appear in this area.

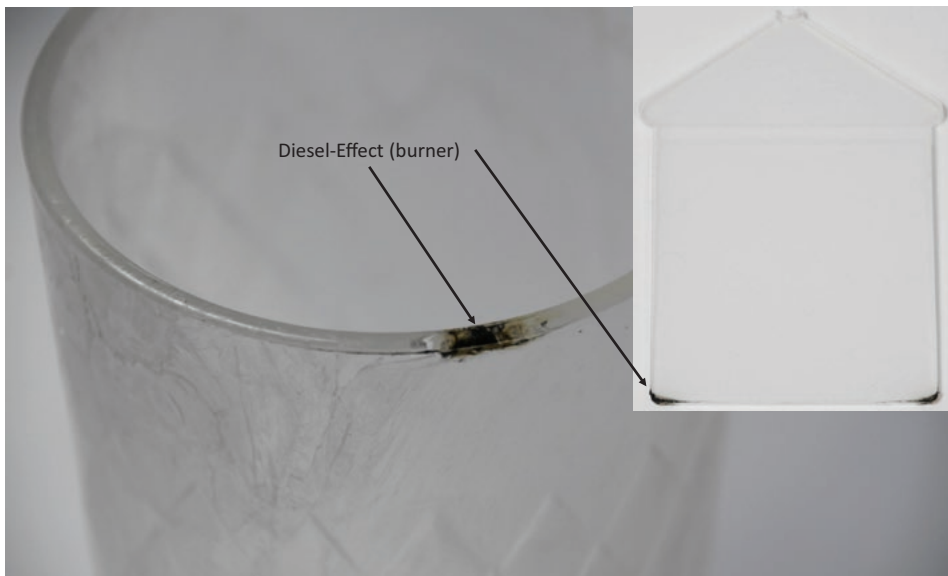


Figure 1.12 Diesel effect (burner)

Possible solutions:

- To avoid the diesel effect, install venting in the mold. Moreover, special inserts for venting, joints and ejectors could be installed to help the air escape.
- Check the venting channels for dirt.
- Optimize the filling process of the molded part accordingly so that air can escape through the joints.
- Lower the melt temperature and the mold temperature to achieve a positive effect.
- Reduce the screw advance speed to decrease the injection speed. Alternatively, it may be sufficient to run a graduated injection profile.
- If the burn spots occur mostly near the parting line, reduce the clamping force of the injection machine.
- All simulation programs indicate the areas where venting should be positioned in the mold. These suggestions can be useful.

1.5.6 Mold Coating (Deposits)

If temperatures are too high during production or if venting is inadequate, a coating can form on the injection mold. The formation of this coating is the result of plastic by-products, or additives, reacting with each other (Figure 1.13). These by-products consist of cracked polymers or other products of decomposition, such as flame retardants.

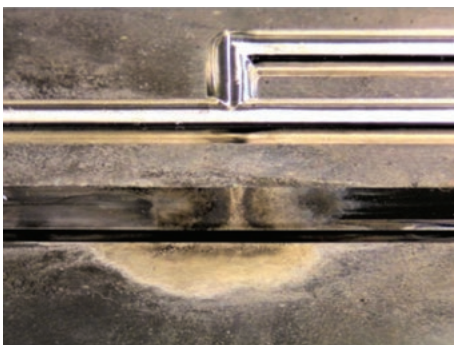


Figure 1.13
Mold coating

Possible solutions:

- Reduce the screw advance speed to decrease the injection speed. Alternatively, it may be possible to run a graduated injection profile, which reduces the speed only in the final phase of filling.

- Compare the gating with the design standards for the sprue size. Make appropriate corrections if execution is poor.
- Compare the melt temperature at the point of exit from the machine nozzle or the hot runner with the recommended values using a probe thermometer. If there are any deviations, adjust the temperature accordingly.
- Check the residence time of the melt in the plasticizing unit and in the hot runner.
- Reduce shearing in the plasticizing unit for a sensitive melt by reducing the screw speed and the back pressure.
- Review the effectiveness and the position of the venting channels and install additional venting channels, if necessary.

1.5.7 Black Specks

Not only thermal damage, which decomposes polymers, but also dirt can cause black specks. These take the form of plate-like or point-like deposits on the surface and the inside of the molded part which are especially visible in transparent materials (Figure 1.14).

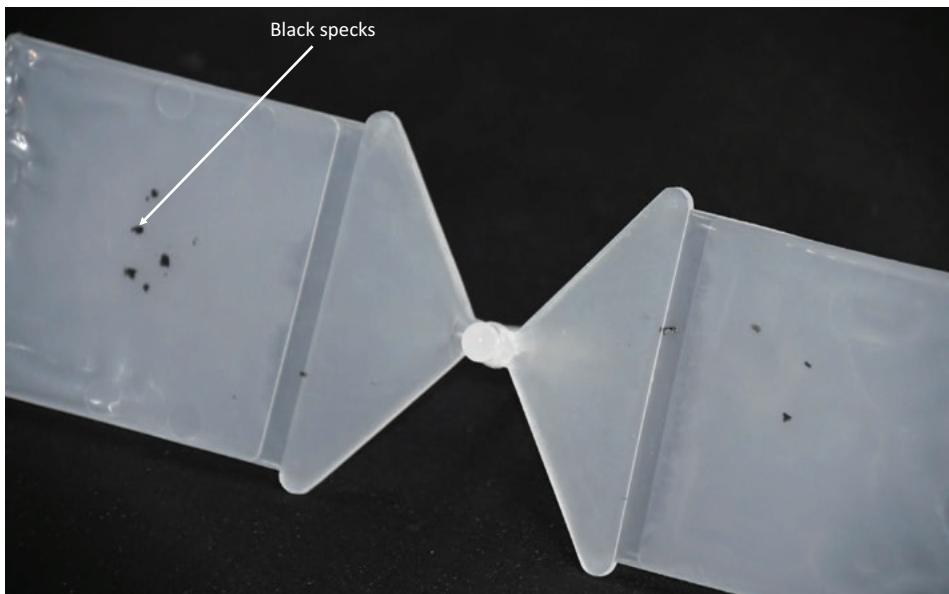


Figure 1.14 Black specks

Possible solutions:

- Check the metering stroke of the injection machine. Optimally, the metering stroke s_D is double the screw diameter D . In general, the following applies: $D < s_D < 3D$. If the metering stroke s_D is less than the screw diameter, the residence time of the melt in the plasticizing unit is too long. Check the residence time of the melt in the hot runner, where present.
- Reduce the melt temperature.
- Reduce the back pressure and/or the speed of the screw.
- If the material needs to be changed, the plasticizing and injection unit as well as the hot runner, if necessary, need to be cleaned thoroughly.
- Avoid dead spots by checking the flow of the melt in the areas of the cylinder and the mold.
- Analyze whether the masterbatch or powdered pigment can be used in combination with the type of plastic that is to be processed.
- Check the functionality of the coating of the mold and/or the plasticizing unit (where present).
- If a material-conveying system is present, check it for abrasion, leaks and dirt deposits.

1.5.8 Ejector Marks

Ejectors can also cause optical damage in the form of differences in gloss on the molded part. Possible faults can show up as differences in gloss, stress whitening (white discolorations) or elevations and indents (Figure 1.15). In some cases, scratches or grooves can occur in the demolding direction. This can be caused by production errors in the mold, such as insufficient stiffness and overly small drafts. Moreover, if the draft angle of the molded part is too small, it may be more difficult to demold a part and this can cause marks. In addition, if the mold is overloaded due to unfavorable production conditions, the molded part can end up being jammed, which also makes demolding more complicated. This applies to high-volume shrinkage as well, which can cause severe shrinkage of molded part onto the core.



Figure 1.15
Ejector marks

Possible solutions:

- Possibly extend the change-over time and/or the post-cooling/residual cooling time.
- Adjust the holding pressure; this generally means reducing it.
- Lower the processing temperatures.
- Treat the cavity with a lubricant or non-stick coating.
- If the ejectors do not match the surface of the mold, make appropriate modifications. However, this only applies in the case of marks and not stress whitening.
- Enlarge the draft angles, if necessary.
- Enlarge any existing ejectors and/or consider further ejectors.
- Strengthen the mold.

1.5.9 Burn Streaks

Severe thermal damage of the plastic melt can lead to burn streaks on the surface of the plastic part (Figure 1.16). The reason is that bubbles develop in the injection molding phase as gaseous fractions of the plastic are released. The bubbles shear at the wall of the cavity, which results in irregular light brown and silver colored, but also dark discoloration. This discoloration can be visible on the surface of the plastic part or in the gating area.

In most cases, the type of discoloration already indicates the root cause of the damage. For example, a long machine standstill while the cylinder heating is still on can often lead to oxidation or disintegration. The severe thermal damage resulting from this becomes visible as light brown to dark brown streaks. In contrast, silver streaks are often the result of strong friction. They are often limited locally to narrow cross-sections of the flow channel or the areas around small nozzles.

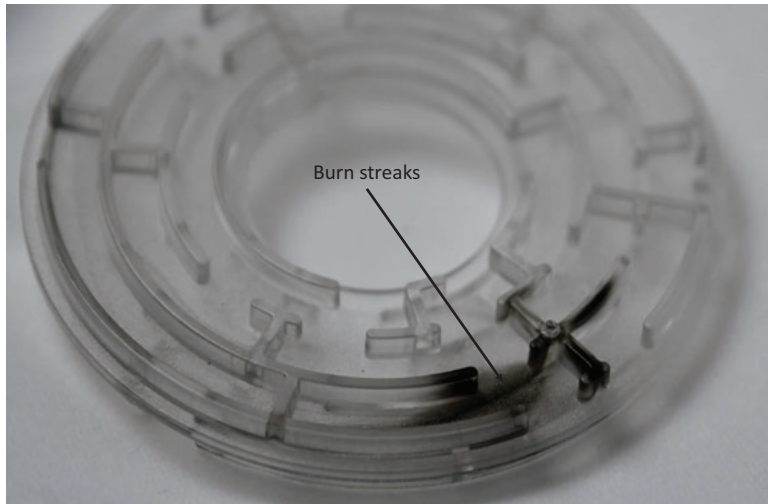


Figure 1.16 Burn streaks

Possible solutions:

- First, check whether the machine was at a standstill before the streaks occurred.
- The plasticizing unit should be checked for its size, that is to say, the residence time. If necessary, swap it out. Twice the screw diameter D is the optimal metering stroke s_D . Generally, $D < s_D < 3D$. If the metering stroke s_D is less than the screw diameter, the residence time of the plasticizing unit is definitely too long.
- Use a probe thermometer to measure the temperature of the melt in the machine nozzle. The temperature should be reduced if it is not in the recommended processing area.
- Additionally, check the temperature of the melt exiting the hot runner. If it exceeds the recommended range, adjust the temperature.
- Reduce the back pressure. Then, evaluate the effects that this change has on the streaks.
- Vary the screw speed to evaluate its impact on the streaks.

- Reduce the injection speed.
- Check the gate geometry and adapt if necessary.
- Avoid sharp transitions and narrow flow cross-section in the cavity, where possible.
- Optimize the flow cross-section of the machine nozzle and/or the hot runner.
- If there is a hot runner, the melt should not remain in it for too long. The residence time can be decreased by reducing the cycle time as a whole. Apart from that, the hot runner needs to be re-dimensioned.

1.5.10 Sink Marks

Sink marks can be another type of surface damage (Figure 1.17). Especially, areas of melt accumulation, such as in the foot of ribs, can be affected. If the design guidelines cannot be maintained when constructing the molded part, local melt accumulations can lead to increased shrinkage in this area. This causes the surface coating to be pulled in. If the surface does not yield, voids can appear instead of sink marks in the molded part.

Often sink marks only develop after the part has been demolded from the mold, as the already cooled edge layers get reheated by the heat coming from the inside of the molded part and they can therefore be deformed again.

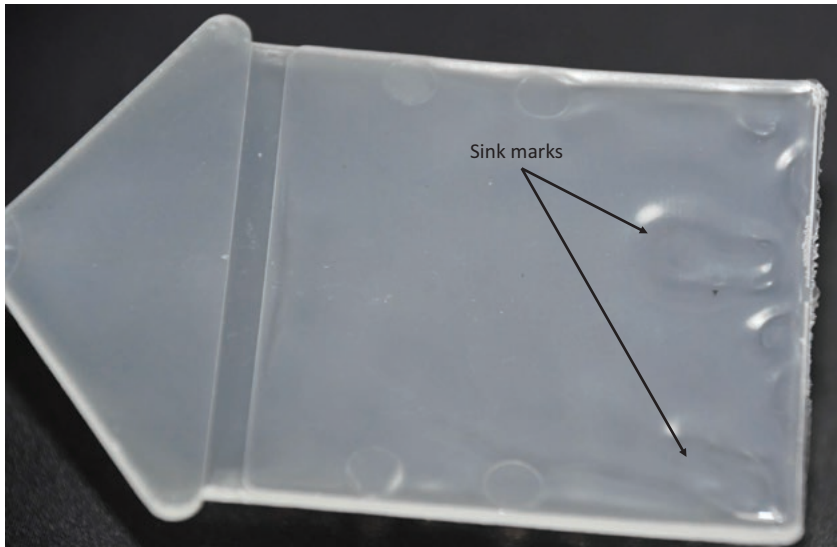


Figure 1.17 Sink marks

Possible solutions:

- To reduce the sink marks, lower the mold and/or the melt temperature.
- Set the holding pressure to a higher setting or, if necessary, extend it. This concerns especially the phase immediately after volumetric filling.
- Alternatively, consider extending the residence time.
- Extend the holding pressure by one or two seconds longer than the sealing time. To calculate the sealing time, the duration is increased in small steps, starting with a short holding pressure time. The weight of the molded part is measured at the corresponding holding pressure time. When the weight remains constant, the sealing point has been reached and the holding pressure can no longer act.
- Reduce the injection speed.
- If the gating point is in an area with a thin wall, consider moving the gate. Doing a simulation in advance could help.
- Enlarge the gate.
- Another measure is to design thinner ribs.
- The screw stroke during injection as well as during application of holding pressure should not sink to zero. Therefore, check the residual melt cushion to ensure that the size or weight of the molded part is not smaller than 3–5 mm.
- Furthermore, check the functionality of the non-return valve. A fluctuating residual melt cushion can indicate a defective non-return valve.
- Cool the molded part once more after the demolding.
- Sink marks can be visualized by the simulation program. The user can therefore examine appropriate measures in advance and adapt them accordingly.

1.5.11 Record Grooves (Cold Meld Lines)

Parallel or concentrated grooves are called record grooves or cold meld lines. They form diagonally to the flow direction and leave marks on the surface of the molded part (Figure 1.18). If the flow rate is too slow, even for a short amount of time, it can lead to cooling of the convex melt front and will result in this effect. The areas of the melt front which are too close to the wall of the mold, cool down too much. Therefore, it cannot be guaranteed that the melt is pressed completely against the mold wall despite the continuously rising pressure and progressive filling. This causes the development of grooves on the surface of the whole melt front.

Another cause can be a temporary melt standstill in front of a thin rib or close to another narrow location. Additionally, changing to holding pressure too early can cause the record grooves effect.



Figure 1.18
Record grooves

Possible solutions:

- Increase the injection speed to reduce the effect.
- Increase the melt temperature, in the hot runner as well, if necessary. The recommended temperature limit should not be exceeded.
- In addition, raise the temperature of the mold, but only to the recommended temperature limit. Increasing the process temperatures also always extends the cooling time.
- The change to holding pressure can be delayed until filling is approximately 98% complete. This may optimize the process.
- Consider changing the location of the gate or the thickness of the wall.

1.5.12 Incomplete Filling of the Mold

If the mold does not fill completely, the molded part may not be fully formed in areas with thin walls or in areas that are distant from the gating point (Figure 1.19). A low dosage (residual melt cushion), not enough pressure, back flow in the return pressure valve or an excessively strong resistance during filling can lead to this effect. The filling resistance consists of the length of the flow path, the wall thickness, and the viscosity.



Figure 1.19 Incomplete filling of the tool

Possible solutions:

- First, check the residual melt cushion and the metering stroke.
- Choosing a more readily flowing material can help to fill the mold better.
- Another option is to increase the melt and/or mold temperature. It is, however, advantageous to first increase the melt temperature, because increasing the relative mold temperature means that the filling pressure cannot be reduced as much. Moreover, increasing the mold temperature can extend the cooling time much more than were the melt temperature to be increased.
- Avoid excessive pressure loss in the gating point and in the cavity.
- Switching from the injection pressure to the holding pressure later can have a positive effect.
- Alternatively, optimize the injection speed. Usually, it needs to be increased.
- Optimize ventilation at the end of the flow path. In the area of the air pocket, install a vent.
- Balance out the filling process.
- Moreover, adjusting the filling pattern by installing flow brakes or aids can lead to an improvement.

- Consider moving the gate. To facilitate, this simulation technology can be used.
- In general, simulation programs can be a useful mold for avoiding incomplete filling.

1.5.13 Color Streaks

If pigments of a powdered pigment, masterbatch or liquid pigment are unevenly homogenized in the plastic matrix, color differences or streaks can occur on the surface of the molded part (Figure 1.20). These streaks occur close to and far away from the gating point and over large areas near the meld lines. They may even occur behind sharp-edged spots. In these areas, pigments accumulate and form color agglomerates, which are responsible for the color differences. The agglomerates near the surface are visible as streaks, whereas the pigment agglomerates in the inner areas of the molded part wall cause patchy color differences.

There are several causes for this molded part flaw: pigments that are not a match for the plasticizing unit, due to an incorrect L/D ratio, or the wrong processing parameters, for example if the melt temperature is not high enough. On the other hand, colored or carrier parts which are incompatible with the plastic to be colored might trigger irregular colored patches or streaks.

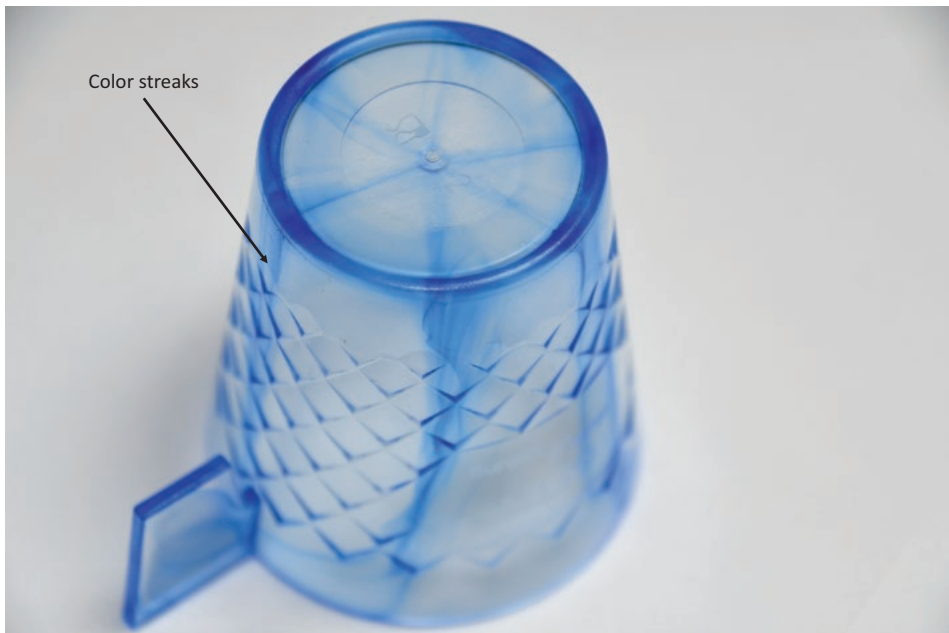


Figure 1.20 Color streaks

Possible solutions:

- First, check the masterbatch data sheet, which is provided by the supplier, for compatibility of the masterbatch with the chosen plastic material. Simultaneously, compare the settings for the masterbatch concentration with those given in the supplier information.
- Moreover, checked whether the temperature of the hot runner is compatible with the masterbatch and whether it falls within the processing range. Heat sensors and the regulation of the hot runner system should be checked for this.
- Increase the injection speed to decrease color streaks.
- Increase the back pressure to help avoid pigment accumulations.
- After increasing or reducing the speed of the screw, analyze the effect on the color streaks.
- To reduce the formation of the color streaks, check whether the melt temperature at the point of entry into the mold is high enough.
- Alternatively, move the gate or adjust the wall thickness.
- Check the plasticizing unit for its size, which means the residence time, or, if necessary, swap it out. The optimal metering stroke s_D is twice the screw diameter D . In general, $D < s_D < 3D$. If the metering stroke is less than the screw diameter, the residence time of the melt in the plasticizing unit is in any event too long. The residence time should be compared against that given in the supplier information. Moreover, the screw specifications (L/D ratio, mix and/or shear component necessary, etc.) should be checked against requirements.

1.5.14 Glass Fiber Streaks

Irregular and rough patches on the plastic mold surface are called glass fiber streaks (Figure 1.21). In some cases, they also occur in regular intervals at the front of the melt. Depending on the angle of the light, they might have a more metallic shimmer or be rather dull. Glass fiber streaks can form especially at turn-arounds, meld lines or breakthroughs, because flow processes near ribs and very thick walls and the processing parameters (mold and melt temperature, injection speed) heavily influence this formation.



Figure 1.21 Glass fiber streaks

Possible solutions:

- Reduce glass fiber streaks significantly by optimizing the homogeneity of the melt. To achieve homogeneity, increase the screw speed and/or the back pressure.
- Increase the mold and/or the melt temperature (if necessary, also in the hot runner). However, keep in mind the recommended temperature limit and the cooling time.
- Increase the screw advance time, if possible, by using a step-by-step injection profile.
- Check whether the gate can be moved.

1.5.15 Moisture Streaks

Another type of streak can occur on the surface of the plastic part: moisture streaks (Figure 1.22). These can form if the residual moisture in the granules or in or on the mold is too high. The latter occurs when moisture is released through leaks in the temperature-control system or through water of condensation. The streaks stand out due to their elongated, parabolic form. However, it should be noted that their tip always points in the direction of the flow path. The moisture causes visi-

ble marks on the surface of the plastic part, as steam bubbles form in the injection phase, which then pop on the melt front. Due to the fountain flow, they get pushed to the surface. Consequently, the bursting water and gas bubbles cause elongated streaks.

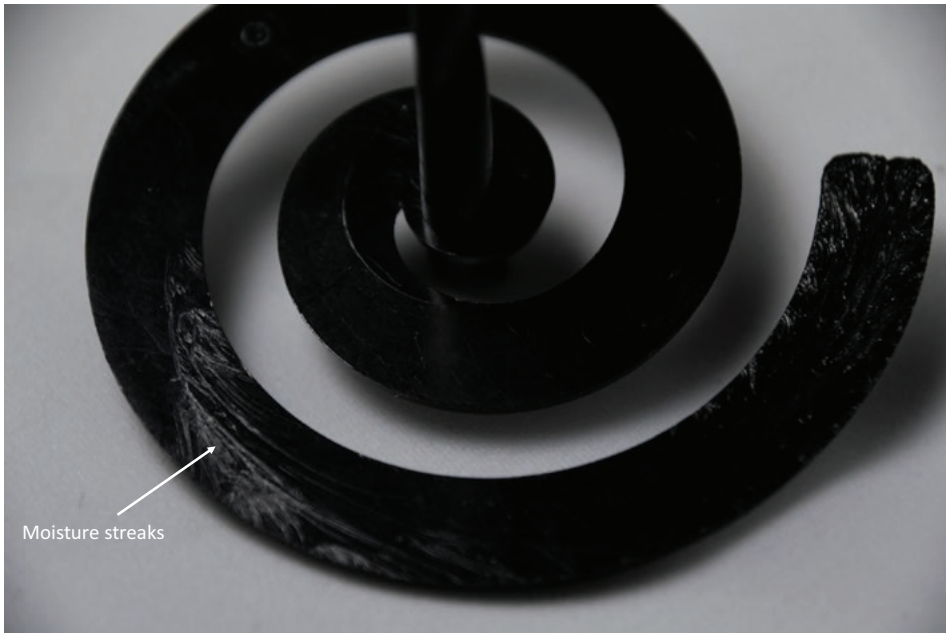


Figure 1.22 Moisture streaks

Possible solutions:

- First, check the granules packaging for external damage.
- Especially in the case of hygroscopic plastics, which tend to absorb water (PA, PET, PC), the residual moisture of the granules must be measured. It is essential to adhere to the maximum allowed level of residual moisture. Otherwise, it may not only cause streaks on the surface but also lead to hydrolytic degradation of polymer chains.
- Reduce the amount of granules in the material hopper.
- Check whether the storage conditions of the granules match the suggestions provided by the raw materials supplier. Furthermore, check the settings for pre-drying.
- Examine the inside of the mold as well. The temperature-control system should not have any leaks and no water of condensation should form on the mold surface.

1.5.16 Delamination

If there is not enough of a connection between the surface layers of a plastic part, delamination can occur. This means that the surface of the plastic part splinters or that it fans out (Figure 1.23). This can also occur when the already cooled edge layer is moved by the shear stress induced by the flow process and therefore peels off. Depending on the type of plastic, this has different impacts on the plastic part. If it is a semi-crystalline polymer, then the layers have different crystal structures. Amorphous thermoplastics, on the other hand, tend to become segregated due to a reaction in the additive/melt/pigment mix.



Figure 1.23
Delamination

Possible solutions:

- Compare the current settings with the settings of the last process that was completed successfully.
- Using the masterbatch data sheet provided by the supplier, check whether the masterbatch is compatible with the chosen plastic material.
- Reduce the injection speed. Simultaneously, increase the manufacturing temperature.
- Moreover, always thoroughly clean the plasticizing unit before a change of materials.
- Avoid sharp turnarounds, especially near the gate, as they can cause the material to shear heavily. Additionally, avoid high wall shear stress (see Chapter 16).

1.5.17 Tiger Stripes

A pulsating melt flow, which is mostly the result of a blend (thermoplastic multi-phase system), can lead to markings diagonally to the flow direction. These markings become visible on the plastic part surface. As the periodic shadows resemble the pattern on the coat of a tiger, they are also called tiger stripes (Figure 1.24). Generally, the elastic features of plastic materials are responsible for this phenomenon. Plastic materials have more or less elastic features. The elastic part functions as a spring and causes the pulsation. This phenomenon is particularly likely to occur when pressure is relieved on the plastic melt and the melt enters the mold in this state (see Chapter 9).

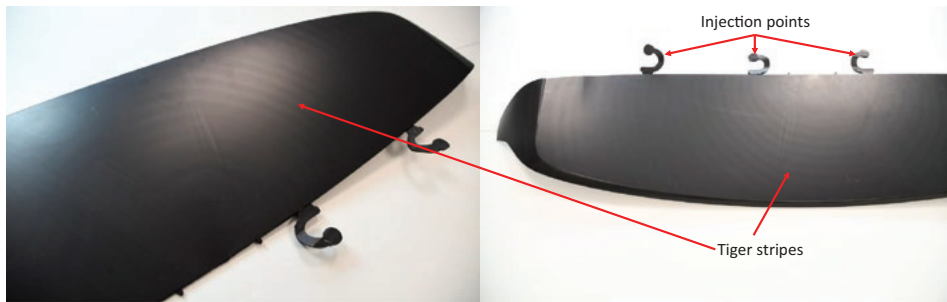


Figure 1.24 Tiger stripes

Possible solutions:

- Enlarge the wall thickness of the plastic part and increase the injection diameter to avoid the occurrence of tiger stripes.
- Moreover, increase the mold and/or melt temperature to help avoid this defect. However, adjust the melt temperature first, as this is often more effective and does not seriously extend the cycle time.
- Furthermore, choose a different material that has better or different flow abilities. In general, a material with less elastic qualities (memory module) can reduce tiger stripes.
- In cascade injection, avoid pressure pulses. Simulation technology can offer a lot of support here.
- Tiger stripes can to a certain extent be visualized by simulation programs. Usually, pressure fluctuations are responsible for this effect, which is why they can be simulated through special hot runner technology. This can aid the finding of possible solutions.

1.5.18 Formation of Stress Cracks, Micro Cracks

If stress inside the material is lower than the tensile stress at break, cracks can occur on the inside and outside of the plastic part. Prior to the occurrence of so-called stress cracks, crazes form in the material (Figure 1.25). Crazes are crack-like, extended areas, which are connected through fibrils, i.e., highly stretched strands of molecules. They tear apart, among other things, when local stress and other tensile stress impact them from the outside. Aggressive media can also be responsible for the cracks. An increase in the notch effect or the spreading and swelling pressure influences the cracks. In addition, the stress within the plastic part, whose strength is impacted by the manufacturing process, also affects the cracks. Due to the cooling processes of the melt in the cold mold, it is not possible to avoid all stress in the plastic part. The aforementioned media, however, can detect the stress caused by the manufacturing process.



Figure 1.25
Formation of stress cracks, micro cracks

Possible solutions:

- Increase the mold temperature without exceeding the temperature limit. It can be particularly advantageous to cool down both walls of the plastic part evenly via the cooling circuit temperature.
- Additionally, reduce the holding pressure.
- Stiffening the mold design can help to avoid micro cracks.
- Generally, avoid melt accumulations and, if necessary, reduce them. For example, the wall thickness of a rib in the foot of the rib can be minimized.
- Stress can also be visualized by the simulation program.

1.5.19 Jetting

With this type of plastic part damage, a strand of plastic melt forms, starting at the gate. It then develops in a snake-like pattern in the cavity and becomes visible in the plastic part (Figure 1.26). Normally during melt flow, the melt is in contact with the wall at the transition point to the large diameter. If, however, the flow rate is too high, the melt will hit the empty space in form of a jet. The melt folds at the point where the jet randomly and briefly strikes the wall of the mold. As soon as the surface of the strand cools, it cannot connect properly with the following melt and so visible damage occurs and the strength of the plastic melt decreases drastically. Jetting cannot be reproduced as it changes shape and spread with each shot. Therefore, it makes no sense to predict jetting with a simulation program. The program always assumes wall adhesion.

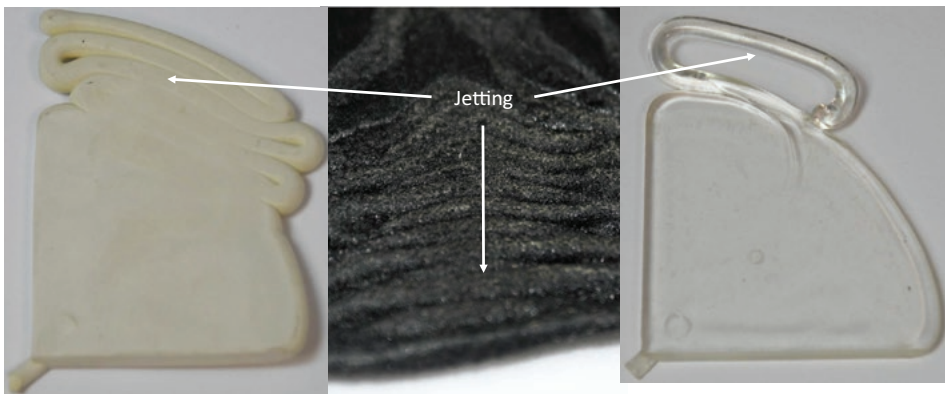


Figure 1.26 Jetting

Possible solutions:

- To avoid jetting, reduce the injection speed.
- Concerning the gate, either enlarge the gate or move it to an area of smaller flow diameter.
- Jetting can be prevented by rounding the transition between plastic part and gate.
- Furthermore, the plastic melt should be faced with an obstacle immediately after the gate.
- As previously established, a jetting cannot be sensibly simulated. On one hand, simulation programs assume wall adhesion, an assumption which contradicts the occurrence of jetting. On the other, jetting cannot be reproduced and can therefore not be simulated.

1.5.20 Voids

After the plastic part has cooled down, microcellular or bubble-like cavities, which are also sometimes called vacuoles, can form inside the plastic part. If one drills into the plastic part in colored water and these do not fill with water, gas bubbles have formed inside the plastic part. However, if it is a truly empty space and water can get in, voids have formed (Figure 1.27). Unlike gas bubbles, voids often occur near the plastic core, which is situated in the middle of the wall or next to melt accumulations. Like sink holes, voids form in areas of high volume shrinkage where melt compression due to the holding pressure is insufficient. Unlike sink holes, voids can also appear when stiff edges are frozen and therefore not flexible anymore. The contraction forces acting as the plastic melt cools inside the part cannot be compensated anymore. As a result, empty spaces form in these areas.

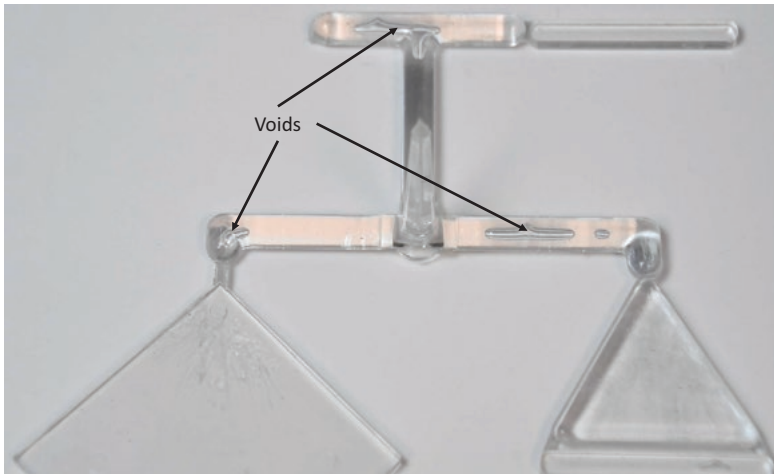


Figure 1.27 Voids

Possible solutions:

- Enlarge the gate. Similarly, move the gate if the gating point is near an area with a small wall thickness. Simulation can be very useful here.
- When designing the ribs, etc., observe the design guidelines.
- The holding pressure time should be one or two seconds longer than the sealing time. To detect the sealing time, a brief holding pressure time is increased in small steps. The weight of the part is measured at each holding pressure time. If the weight remains constant, it means that the sealing point has been reached and that the holding pressure is no longer acting.
- Another solution would be to increase the holding pressure.

Index

Symbols

3D-F meshing 73
3D-F process 73
3D measuring technology 354
3D point clouds 356
3D RSC orientation model 484
3D-V process 76
96-cavity mold 206

A

air entrapment 434
air pockets 36, 148, 150, 153, 586, 587
alignment 621
alternating loads 412
amorphous polymer 509
Ampcoloy 301, 322, 336, 445, 448
analysis of variance 529
analysis of variance tables 529
angle of incidence 377, 380, 409
anisotropic non-linear material description 496
anisotropic properties 302, 463
ANOVA-Tabellen 529
aspect ratio 80
Assembly Design 680
a surface model 358
auxiliary axis system 621
axial stress 499

B

balance equations 59
balancing 185, 186
– mathematical rheological 187, 192, 196, 197, 212

– natural rheological 186, 208
– thermal 211, 212
barrier property 562
basic exercises 105
beams 73
best-fit 356, 359
Biot number 276
black specks 18
BLM mesh 399
BLM method 72
bodies 602
Boolean operations 614, 676
boundary layers 399, 493
braking effect 129
breaking stress 397, 402, 404
breakthrough 121
breakthrough in the molded part 378
Brinkman number 158
buoyancy forces 419
burners 16
burn streaks 20
burr formation 191

C

calculation time 76, 662
calibrating 87, 502
Carreau-WLF model 90, 474
cartridge heaters 592
cascade 225, 237
cascade control 199
cascade injection molding 240, 242, 251, 252, 424
Cauchy stress tensor 62
cause-and-effect model 519
cavity 5

- cavity plate 5
 - cavity pressure 9, 411, 507, 512, 514
 - curve 166, 507, 512
 - sensor 178, 507, 512, 516
 - transducer 510
 - centering 455
 - center layer 470, 472, 500
 - central setup 371
 - clamping force 10, 190, 195, 458, 569, 573, 574
 - clamping points 441
 - closure formulation 482
 - coating 17
 - coefficient of linear thermal expansion 98
 - cold runner 105, 107, 170, 172
 - system 589
 - technology 582
 - collision 676, 677
 - collision analysis 604, 680
 - color streaks 26
 - component quality 564
 - compression line 580
 - compression phase 8, 9
 - computational effort 601
 - computer tomography 481
 - confidence interval 547
 - confirmation experiment 525
 - conservation equations 477
 - contact condition 680
 - control variables 517, 521
 - control volume 60
 - coolant temperature 334
 - cooling 325
 - near-contour 321
 - cooling behavior 300
 - cooling channels 6
 - cooling equation 167
 - cooling error 275
 - cooling medium 6
 - cooling module 61
 - cooling phase 11, 167
 - cooling process 278
 - cooling rates 94, 295
 - cooling system 311
 - cooling time 167, 168, 275, 279
 - copper alloys 301, 322, 336
 - copper pins 336, 338
 - core 5, 439, 458, 459
 - core component 555, 564, 575, 577
 - core deformation 440
 - core layer 559
 - core offset 440, 446, 455, 459
 - correction factor 620
 - counteract 227
 - coupling 629
 - critical shrinkage 311
 - crosslinking molding compounds 579
 - cross-section 260
 - annular 108, 110
 - flow 108, 109
 - Cross-WLF model 89, 515
 - crowning 227
 - crystallinity 352
 - crystallite melting region 120
 - crystallization behavior 386, 509
 - curing
 - degree of 590
 - speed of 582
 - curing kinetics 583, 584
 - curing reaction 579
 - curing time 589
 - cycle time 3, 352
 - cycle time optimization 590
- D**
- D2-type shrinkage plate 303
 - dashboard 547, 552
 - data exchange 670
 - data fitting 100
 - data transfer 602
 - data volume 642
 - deflection 413
 - deformation 14, 320, 324, 370, 371, 412, 415, 416, 431, 433, 439, 450
 - deformation analysis 353
 - deformations 533
 - degradation 218, 234
 - degree of cooling 165, 280
 - degree of freedom 529
 - delamination 30, 584
 - delay time 573
 - demolding aids 387
 - demolding direction 458
 - demolding shrinkage 295
 - demolding temperature 96, 167, 269, 275, 279, 328, 330, 334, 338, 340, 342, 344, 373, 507

- average 167, 279
- maximum 167, 168, 280
- design elements 668
- design of experiments 403
- Design of Experiments (DoE) 519, 591
- detailed areas 600
- detailed geometries 601
- deviation
 - mean squared 529
- diesel effect 16, 148, 153, 392
- dimensional accuracy 508
- direction of stress 393
- discretization 45, 65, 70, 491, 562, 576
- display accuracy 604
- dissipation 140, 406
- disturbance variables 521
- DoE 540, 552, 585
- dual domain analysis 68
- dwelt time 185, 230
- dynamic feed 255

E

- economic efficiency 347, 564
- edge effects 146
- edge layers 78, 468, 471, 499, 580
 - amorphous 352
 - frozen 107, 115, 119, 145, 159
 - sheared 177, 258
- efficiency 269
- ejection temperature 3, 88
- ejector box 413
- ejector plate 6
- ejector side 6
- elastomers 579
- element normal method 75
- elongational flow 260, 467, 469, 470, 479, 557
- elongational viscosity 261, 564
- entanglement 384
- equivalent stress 431, 432, 445, 450
- error estimation 530
- Eulerian model 478
- evacuation 582
- exchange formats 55
- experiment 522, 537
- experimental designs
 - full factorial 545
 - partial factorial 545

- experimental plan 536
- export preparations 672
- extrapolating 631

F

- factors 519
- family molds 183, 184, 188, 197
- fatigue loading 445
- FEA interface 486
- fiber breakage 393
- fiber detachment 393
- fiber interaction coefficient 482, 483
- fiber length analysis 501
- fiber orientation 399
- fiber orientation file 491
- fillet 644
- filling behavior 220
- filling module 56
- filling pattern 152, 226, 240
- filling phase 59
- filling pressure 116, 177, 218, 224
- filling simulations
 - fast 54
- filling time 158, 237, 513
- film gate 470
- finger effect 561, 562
- finite-difference method 62
- finite-element method 63
- finite elements 70
- finite elements calculation method (FE) 477
- finite-volume method 63
- fitting 56, 87, 451, 502
- fitting process 101
- fixing 457
- flash 35
- flatness 324
- flow brake 379
- flow characteristics 164
- flow direction 464
- flow forms 479
- flow front 123, 125, 466
- flow front speed 158
- flow instabilities 558
- flow path 112, 472
- flow-path/wall-thickness diagram 112, 114, 159, 217, 228, 421
- flow resistance 120, 127, 201, 258, 568
- flow simulation 465, 477, 490

flow spiral 103, 159, 162, 218, 512
 flow temperature 282
 foaming 374
 Folgar-Tucker equation 479
 Folgar-Tucker model 479, 484
 food approval 562
 formation of the molded part 8
 fountain flow 50, 70, 381, 386, 397, 466
 free edge 80
 freezing time 314
 full factorial matrix experiment 523
 functional reliability 411
 fusion 68
 F-value 529

G

gaps 618, 623, 630, 631, 633
 gas
 – introduction of 567, 572
 gas-bubble spread 572
 gas-feed channel 567, 571
 gas injection molding 555, 564
 gas injection process 222, 228
 gating point 224, 228, 379, 389
 gating position 585
 gating system 169
 G.B. Jeffery 478
 Generative Shape Design 619, 620, 643, 676
 geometry changes 601
 glass fiber 302, 310, 384, 393, 463, 501
 glass fiber content 395, 474, 483, 564, 565
 glass fiber length 396, 503
 glass fiber orientation 394, 398
 glass fiber streaks 27
 glass temperature 89
 gloss differences 11
 GOM 353
 GOM Inspect 356

H

Hagen-Poiseuille 110, 112, 119, 120, 158, 196, 516
 heat balance 189
 heat-conducting inserts 348
 heat conduction 279
 heat dissipation 322, 445
 – inhomogeneous 276

heat exchange 279
 heat flow balance 269
 heat flux 300
 heating cartridges 583, 591
 heating phase 596
 heat transfer coefficient 276, 281
 Hele-Shaw flow 67, 70
 high-pressure capillary rheometer 163
 history 626, 673
 holding pressure 2, 234, 303, 314, 509, 570
 holding pressure module 57
 holding pressure phase 4, 10, 164, 508, 509, 511, 512, 573
 holding pressure time 157, 165
 hot runner 105, 107, 169, 174, 185, 209, 219, 229, 413, 677
 – externally heated 174
 – internally heated 174
 – open 178
 hot runner nozzles 225
 hot runner system 178
 – heat balance 177
 – internally heated 109
 – thermal and rheological design 176
 hotspot 182, 290, 313, 328, 332, 334, 338, 340, 342, 345, 360, 374
 Hybrid Design 607

I

IKV model 94
 incomplete filling of the mold 24
 influencing factors 519
 inhibitor 583
 initial length 501
 injection modules 567
 injection mold 5, 411
 – load 425
 injection molding cycle 2
 injection molding machine 1
 injection molding process 3
 injection phase 4, 8, 158, 511
 injection pressure 221
 injection sequence 96
 injection speed 116
 injection time 116
 inlet pressure losses 262
 insert 379, 429
 instabilities 562

Institute for Plastics Technology in Darmstadt
ikd 496
integrative simulation 398, 465, 485
interaction 390, 391, 523, 531, 540
interaction term 479
internal pressure 499
intersections 81
intrinsic viscosity 48
isochrone curve 164
isochrones 148, 254, 255

J

jaws 230, 414, 424
jets 33, 51
jetting 70
joining 628

K

Kessel formula 499
key figures 516, 517
knit line 12, 121, 150, 220, 224, 229, 230,
232, 243, 251, 312, 350, 377, 399, 492, 494
– different types 380
– stagnant 380
knit line factor 391, 397
knit line quality 123, 129, 147, 148, 382, 383,
390, 404, 498
knit line strength 383, 386, 391, 395, 397,
398, 402

L

Lagrangian model 477
laminar layer flow 466
Laplace equation 61
laser melting 347
layer boundary profile 557
layered laminar flow 51
lettering 652
levels 522, 536, 544
limiting shear stress 405
liquid silicone rubber (LSR) 579
liquid silicones 579
load calculation 440
locking force. *See* clamping force
long glass fibers 326, 396, 503
LSR injection molding 582

M

machine parameters 520
macromolecules 384
manifold edge 80
manifold systems 201
manipulated variables 521
mapping 400, 422, 496
mapping program 465, 485, 489, 502
material card 56, 87, 162, 165, 218, 514, 584
– rheological 89
– standard 100, 103
– thermal 92
material characteristics 464
material laws 60, 496
material map 487, 502
– non-linear 465
material model 402, 489, 491
– linear-elastic 402
material stress 116
matrix 463
matrix equations system 63
matrix experiment 526
matrix fracture 393
mean value 527
– arithmetic 527
measuring distances 354
mechanical design 415, 465
mechanical properties 381, 394, 402, 439,
445, 460, 463, 464, 487, 502
meld lines 122, 312, 377
melt accumulation 314, 316, 373
melt flow front 213, 214, 216
melt load 421
melt pressure
– specific 159, 161, 163, 165
melt temperature 234, 245
melt volume rate 386
mesh density 65, 398
mesh errors 80
meshing 55, 65, 66, 73, 490, 576, 599
– types of 67, 72
meshing problems 600
meshing quality 82
meshing technologies 65
midplane analysis 67
midplane mesh 67
midplane model 144
model equation 523

moisture streaks 28
 mold 1
 mold cooling 318
 mold damage 191
 mold deformation 413, 423, 424, 440
 mold dimensioning 40
 molded part characteristics 516
 molded part faults 11
 molded-part formation 157
 molded part properties 508, 516
 molded part quality 507
 molding accuracy 350
 molding process 294, 507
 mold loading 417, 423
 mold surface temperature 272
 mold temperature 282, 328, 332
 molecular entanglement 388
 more product properties 521
 morphology 352
 multi-cavity molds 183, 184, 188, 200
 multi-point gating 219, 220, 228
 multipurpose bodies 492

N

Navier-Stokes equations 59
 Navier-Stokes model 76
 near-contour cooling 345, 348–351, 374
 needle valve 105, 175
 needle valve nozzle 589
 Newton's law of friction 48, 516
 node normal method 75
 no-flow temperature 88, 129, 133, 143, 151, 239, 434
 non-compressed area 580
 non-manifold edge 80
 notches 377, 382, 384
 nozzle side 6
 nozzle temperature 211
 nozzle tip 175
 nucleating agent 102
 Nusselt number 276

O

offset 443, 450
 opening 418, 422
 operating point dependence 196
 optimization 543, 598

optimization goal 547, 552
 orientation 306, 311, 326, 378, 381, 384, 394, 464–470, 479–481, 499, 634
 – degree 482
 orientation dynamics 481
 orientation tensor 57, 394, 471
 original geometry 601
 orthogonal fields 524
 Ostwald and de Waele 475
 overlapping 81

P

parameter 540
 parent elements 626
 Pareto chart 369
 Part Design 622, 640, 682
 partial factorial experiment 523
 phases 7
 piezoelectric sensor 507
 pigmented material 102
 plasticizing unit 501
 plate thickness 416
 plug 38
 Poisson's equation 61
 polygonization 356
 polysurfaces 623, 629, 651
 polysurface structure 636
 positioning 607, 619, 621
 positioning errors 680
 post-processor 45, 54
 post-shrinkage 296, 298
 power-law model 475
 Prandl number 277
 pre-filling
 – degree of 559, 560, 569, 573
 pre-processing 45
 pre-processor 54
 pressure consumer 177
 pressure control 256
 pressure curve 573
 pressure drop 185, 193
 pressure fluctuations 241, 250
 pressure gradient 430, 432, 443, 452, 457
 pressure limit 114
 pressure loss 105, 109, 186, 197, 203, 237, 238, 260, 266
 – in the distribution system 41
 – in the machine nozzle 41

pressure requirement 209, 216, 229, 234, 246, 586
pressure requirements 190
pressure sensors 580
principal normal stress 422
process control 517
– variothermal or dynamic 349
process data 88
processing shrinkage 296, 298, 303, 549
process optimization 44, 517, 519
process parameters 519, 520, 552
process phases 507
product design 386
product development 44
product properties 520
properties
– external and internal 519
pseudo-plasticity 49, 257, 467, 472, 475
pvT behavior 57, 514
pvT diagram 96, 165, 509
pvT parameter 94

Q

quality 195, 269
quality characteristics 507, 517, 533, 535, 537, 538, 543, 547, 554, 585, 586
quality feature 591
quality management 524
quasi-steady state 271

R

radial runout 323
radii 645
randomization 526
reciprocal transition shear rate 90
record grooves effect 23
Reduced Strain Closure (RSC model) 481
reference temperature 90
reinforcing materials 386
release agents 387
Renner model 94
residual cooling time 3, 5, 157
residual moisture 387
residual stresses 191, 293, 437
residual wall thickness 439, 564–566
residual wall thickness formation 564
return temperature 282, 322

Reynolds number 47, 281, 290
rheological mold design
– computer-aided 43
ridges 35
rigidity 411, 415
rigidity matrix 465
riser cooling 330, 334, 345
robust design 520, 526
rushing effect 460

S

sandwich injection molding 555
sandwich structure 555
scatter 528
sealing point 2, 10, 165, 509
secondary runner 205
section
– long 635
– short 635
semi-crystalline polymer 509
sensors 211, 213, 214, 225, 237, 240, 243, 248, 510, 597
setting parameters 1, 388, 390, 544
setup parameters 88
set value 548
shear 201, 202
shear flows 557
shear heating 460
shear loads 501
shear rate 48, 88, 192, 257, 263, 265, 466, 467, 475, 516
shear strength 437
shear stress 88, 89, 126, 129, 132, 139, 140, 143, 192, 202, 405, 467, 565
shear stress profile 49
shear warming 129
sheets
– with reinforcement ribs 133
shell elements 144
shell models 77
shift 451
shrinkage 292, 293, 298, 310, 323, 371, 533, 541
– difference 300
shrinkage and warpage behavior 353
shrinkage compensation 314
shrinkage differences 307, 311, 313, 328
shrinkage plate 309

shrinkage type 298
 shut-off needle 175
 shut-off needles 226, 241, 266
 signal-to-noise ratio 526, 527
 simplification 603
 simplification potential 601
 simulation
 – structural mechanics 497
 simulation calculation 41
 – results 53
 simulation model 55
 simulation programs 6, 42
 single part format 611
 sink marks 22, 573
 Six Sigma 524
 skin component 555, 575, 577
 sliders 230, 538, 547, 586
 slippery film 580
 smell 40
 solid 618
 solid body 612
 solid model 640
 special processes 555
 specification tree 605, 612
 specific heat capacity 92, 270
 specific melt pressure 514
 specific volume 95
 speed gradient 467
 speed profile 472, 564
 sphere method 75
 split points 387
 spots 39
 sprue gate 169, 172, 173, 232
 stagnation 132, 139, 143, 150, 151
 standard climate 355
 standard cooling system 327
 statistical design 519
 statistical design of experiments 543
 statistical process models 520, 521
 steel core 441
 stick-slip effect 581
 strain rate 466
 streaks 350, 351
 strength 377, 384, 499
 strength of materials theory 413
 strength properties 565
 strength value 496, 498
 stress cracks 32
 stress peaks 420, 427

stress-strain diagram 401, 464, 465, 487,
 488
 stress whitening 19
 structural analysis 398, 465, 502
 structural simulation 477, 485, 490, 504
 support 413, 435
 surface boundaries 605, 629
 surface comparison 358
 surface differences 365
 surface markings 253
 surface model 618, 640
 surface quality 254, 508, 516
 surfaces
 – neighboring 636
 switching point 8

T

Taguchi 534
 Taguchi experiment 524, 531
 Taguchi method 536
 Tait model 94
 tangential stress 499
 target variables 519, 540
 temperature 239
 temperature constant 407
 temperature control channel 274
 temperature control circuit 317, 318, 327
 temperature control error 277
 temperature control medium 271, 336
 temperature control medium flow 270
 temperature control system 179, 269, 282,
 300, 316
 temperature difference 135, 271, 272, 276,
 284, 286, 288, 290, 330, 593
 temperature profile 278
 temperature sensor 211, 516
 tensile stress 464, 465, 488
 tensile test 464, 488, 492
 tensor components 484
 test method 390
 test plan 317, 368, 585, 591
 the clamping force 411
 the pre-filling level 574
 thermal buildup 596
 thermal characteristic values 514
 thermal conductivity 92, 301, 439, 445, 458,
 460, 597
 – effective 92

thermal cooling process 159
thermal expansion 678
thermal image 180
thermal isolation 175, 182, 267
thermally conductive mold inserts 349
thermosets 579
thickness moment coefficient 482
thin spot 121, 125, 127, 132, 138, 147
– close to the sprue 123
– far from the sprue 133
three-plate mold 173
tiger stripes 31, 253
tolerance 543
tolerance range 547
topological errors 628
total cooling time 157, 165
total displacement 356, 361, 364, 365
transition region 96
transition temperature 96
transverse contraction 98
transverse flow 468
triangle aspect ratio 82
trimming 650, 651
tunnel gate 172, 173, 441
turbulent flow 281
turnarounds 263, 501

U

unsteady state 271
unwarp 227, 372
usability 603

V

vacuum 591
vacuum soldering process 347
valve gate 176
valve gate nozzles 229
valve gate system 107, 218, 237, 568
variables 535
variance 520, 526
Varimos 317, 540, 552–554
velocity 48
velocity curve 48
velocity gradient 49
venting 153, 154, 382, 388, 390–392, 401,
492, 586
vents 590

viscoelastic effects 60
viscoelastic flow behavior 253
viscoelastic flow properties 250
viscosity 47, 163, 214, 229, 258, 263, 265,
350, 386, 516, 575, 582, 590
viscosity curve 90, 579
viscosity exponent 89
viscosity ratio 558, 559
voids 34, 293, 373, 460
volume contraction 295
volume meshing 69, 73
volume model 69
volume shrinkage 10, 97, 294, 300, 303,
307, 313, 374, 569, 573

W

wall adhesion 51, 466, 467, 580, 581
wall shear rate 516, 517
wall shear stress 218, 516
wall slipping 580
wall temperature 516
wall thickness distribution 460
warpage 227, 235, 249, 275, 284, 288, 293,
307, 311, 318, 322–324, 371, 542, 550
warping 15
warping calculation 62
water absorption 404
weak point 380
wear 412
welding process 381, 397
weld line. *See* knit line
Williams, Landel and Ferry model 90
WLF model. *See* Williams, Landel and Ferry
model

Y

Young's modulus 98, 501

Z

zero point 620, 671
zero-shear viscosity 89, 90