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

Pre	face		XV
For	eword		XVII
The	Author		XIX
1	Basics	of Injection Molding	1
1.1	Compo	nents of an Injection Molding Machine	1
1.2	The Inj	ection Molding Cycle	2
1.3	The Inj	ection Mold	5
1.4	The Pha	ases 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	Injectio	n Molding Simulation	40
	1.6.1	Simulation Calculation – Typical Methods	45
2	Rheolo	gical Basics	47
3	Introdu	action 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	Mathe	matical Basics	59
4.1	Calcula	tion Phase of Injection Molding Simulation	59
4.2	Calcula	tion Variations	62
	4.2.1	Finite-Difference Method (FDM)	62
	4.2.2	Finite-Volume Method (FVM)	63
		Γ_{i} : (t_{i}, Γ_{i}) (t_{i}, t_{i}) (t_{i}, τ_{i})	()

5	Types	of Meshi	ng, Errors in Meshing and Meshing Quality	65
5.1	Types	of Finite E	lements	65
	5.1.1	Moldflow	v 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	Differen	ce 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 Meshir	ng on Simulation Results	76
5.3	Possibl	le Mesh Ei	rors	80
	5.3.1	Free Edg	ges	80
	5.3.2	Overlap	ping	81
	5.3.3	Intersec	tions	81
	5.3.4	Aspect F	Ratio	82
5.4	Evalua	tion of the	Meshing Quality	82
6	Materi	al Cards	in Simulation Programs	87
6.1	Setup a	and Proces	ss Data	88
6.2	Rheolo	gical Mate	erial Card	89
6.3	Therm	al Materia	l Card	92
6.4	Therm	odynamic	Material Properties – pvT Diagram	94
6.5	Mecha	nical Prop	erties	98
6.6	Creatir	ng a Mater	ial Card	100
7	Easy B	asic Exe	rcises for Simulating Injection Processes	105
7.1	Compa	rison of S	prue Systems in Terms of Pressure Requirement	105
7.2	Flow-P	ath/Wall-1	hickness Diagram	112
7.3	Calcula	ating the (Optimal Process Window during Injection	116
7.4	Sheet v	with Break	through – Filling Behavior	121
7.5	Sheet v	with Trans	verse Flow Stream toward the Rib	133
7.6	Sheet v	with Two I	Flow Directions and One-Sided Thin Spot	138

7.7	Paralle	Parallel Flow Paths of Different Widths 14			
7.8	Sheet with Central Thin Spot 1			147	
7.9	Box wi	th Thick H	Rim	151	
8	Simula Injecti	ating the on Mold	Molded-Part Formation Process in the	157	
8.1	Injectio	on Phase		158	
	8.1.1	Detectio	on of the Optimal Injection Time and Speed	158	
	8.1.2	Compar Simulat	ison of the Real Injection Pressure with the ed Injection Pressure Based on a Flow Spiral	159	
		8.1.2.1	Flow-Path/Wall-Thickness Diagram – Comparing Reality and Simulation with the Standard		
		8.1.2.2	Material Card Flow-Path/Wall-Thickness Diagram – Comparing Reality and Simulation with the Customer	161	
			Material Card	163	
8.2	Holdin	g Pressur	e Phase	164	
8.3	Cooling	g Phase .		167	
9	Gating	System	s in Injection Molding	169	
9 9.1	Gating Genera	; System al Informa	s in Injection Molding tion on Gating Systems	169 170	
9 9.1 9.2	Gating Genera Cold R	; System ıl Informa unner Sys	s in Injection Molding tion on Gating Systems tems for Thermoplastics	169 170 172	
9 9.1 9.2 9.3	Gating Genera Cold R Hot Ru	; System ıl Informa unner Sys nner Syst	s in Injection Molding tion on Gating Systems etems for Thermoplastics ems	169 170 172 174	
9 9.1 9.2 9.3	Genera Genera Cold R Hot Ru 9.3.1	; System Il Informa unner Sys nner Syst Simulat	s in Injection Molding tion on Gating Systems etems for Thermoplastics tems ion of Single-Point Gating with Open Hot Runner	169 170 172 174 178	
9 9.1 9.2 9.3	Genera Cold R Hot Ru 9.3.1 9.3.2	s System Il Informa unner Syst nner Syst Simulat Multi-Ca	s in Injection Molding ation on Gating Systems atems for Thermoplastics tems ion of Single-Point Gating with Open Hot Runner avity Molds	 169 170 172 174 178 183 	
9 9.1 9.2 9.3	Genera Cold R Hot Ru 9.3.1 9.3.2	System Il Informa unner Syst Simulat Multi-Ca 9.3.2.1	s in Injection Molding tion on Gating Systems terms for Thermoplastics terms ion of Single-Point Gating with Open Hot Runner avity Molds Balancing Methods for Multi-Cavity Molds	 169 170 172 174 178 183 186 	
9 9.1 9.2 9.3	Genera Cold R Hot Ru 9.3.1 9.3.2	s System al Informa unner Syst Simulat Multi-Ca 9.3.2.1 9.3.2.2	s in Injection Molding tion on Gating Systems terms for Thermoplastics terms ion of Single-Point Gating with Open Hot Runner avity Molds Balancing Methods for Multi-Cavity Molds Mathematical Rheological Balancing of Family Molds	 169 170 172 174 178 183 186 197 	
 9.1 9.2 9.3 	Genera Cold R Hot Ru 9.3.1 9.3.2	s System al Informa unner Syst Simulat Multi-Ca 9.3.2.1 9.3.2.2	s in Injection Molding tion on Gating Systems items for Thermoplastics ems ion of Single-Point Gating with Open Hot Runner avity Molds Balancing Methods for Multi-Cavity Molds Mathematical Rheological Balancing of Family Molds Natural Rheological Balancing of a Multi-Cavity Mold	 169 170 172 174 178 183 186 197 200 	
99.19.29.3	Genera Cold R Hot Ru 9.3.1 9.3.2	s System al Informa unner Syst Simulat Multi-Ca 9.3.2.1 9.3.2.2 9.3.2.3 9.3.2.4	s in Injection Molding tion on Gating Systems terms for Thermoplastics terms ion of Single-Point Gating with Open Hot Runner avity Molds Balancing Methods for Multi-Cavity Molds Mathematical Rheological Balancing of Family Molds Natural Rheological Balancing of a Multi-Cavity Mold Thermal Balancing of a 2+2 Family Mold	 169 170 172 174 178 183 186 197 200 	
99.19.29.3	Genera Cold R Hot Ru 9.3.1 9.3.2	s System Il Informa unner Syst Simulat Multi-Ca 9.3.2.1 9.3.2.2 9.3.2.3 9.3.2.4	s in Injection Molding	 169 170 172 174 178 183 186 197 200 211 211 	
9 9.1 9.2 9.3	Gating Genera Cold R Hot Ru 9.3.1 9.3.2	s System Il Informa unner Syst Simulat Multi-Ca 9.3.2.1 9.3.2.2 9.3.2.3 9.3.2.4 Large M	s in Injection Molding	 169 170 172 174 178 183 186 197 200 211 216 	
9 9.1 9.2 9.3	Genera Cold R Hot Ru 9.3.1 9.3.2	s System al Informa unner Syst Simulat Multi-Ca 9.3.2.1 9.3.2.2 9.3.2.3 9.3.2.4 Large M 9.3.3.1	s in Injection Molding	 169 170 172 174 178 183 186 197 200 211 216 219 	
9 9.1 9.2 9.3	Genera Cold R Hot Ru 9.3.1 9.3.2	System al Informa unner Syst Simulat Multi-Ca 9.3.2.1 9.3.2.2 9.3.2.3 9.3.2.4 Large M 9.3.3.1 9.3.3.2	s in Injection Molding	 169 170 172 174 178 183 186 197 200 211 216 219 240 	

	9.3.5	Pressure	e Losses at Abrupt Changes in Cross-Section	259
	9.3.6	Pressure	e Losses at Baffles	263
	9.3.7	Simulati	on of Valve Gate System	266
10	Tempe	rature C	ontrol Systems for Injection Molds	269
10.1	Exampl	le of Temp	perature Control	282
10.2	Factors	Influenci	ng Shrinkage and Warpage	292
	10.2.1	Influenc Shrinka	e of the Temperature Control System on ge and Warpage	300
	10.2.2	Influenc	e of Fillers on Shrinkage and Warpage	302
	10.2.3	Compari	ison of Simulated Shrinkage Values with Reality	309
	10.2.4	Simulati Molded	on of Shrinkage and Warpage Using Sample 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 For	mation during Injection Molding	373
11	Knit Li	nes and	Meld Lines	377
11.1	Format	ion of Kni	t Lines and Meld Lines	378
11.2	Factors	rs Influencing the Quality of the Knit Line		
11.3	Effects Molded	of the Knit Line on the Mechanical Properties of the		
114	Factors	Influenci	ng the Knit Line Strength	386
11.1	11 4 1	Influenc	e of Setting Parameters on Knit Line Strength	390
	11.4.2	Influenc	e of Venting on Knit Line Strength	392
	11.4.3	Influenc	e of Glass Fiber Content on Knit Line Strength	393
11.5	Simula	tion of Kn	it Line Strength	398
11.0	11.5.1	Compari	ison of the Simulated Strength of a Tensile Bar	402
11 /	Dave and 1	with Kn		403
11.0	Exampl	les of Knii	Lilles	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 Co	onsole	424	
		12.1.2.1	Center Console with Hot Runner	424	
13	Simulat	tion of In	serts	429	
13.1	Simple	Basic Exan	nple of a Simulation	429	
13.2	Simulat	ion of a Ba	ack-Molded Insert Made of Cast Iron	436	
14	Core Of	fset Sim	ulation	439	
14.1	Example	es of Core	Offset Simulation	439	
	14.1.1	Push-Butt	ton	441	
	14.1.2	Automotiv	ve Fitting	451	
	14.1.3	Pump Ho	using	457	
15	Integra	tive Simu	Ilation – Structural Analysis	463	
15.1	Formati	on of Orie	ntations during Injection Molding	466	
15.2	Eulerian and Lagrangian Models 477			477	
15.3	Materia	l Models fo	or Orientation Calculation	478	
15.4	The Process of Integrative Structural Simulation			485	
15.5	Simulation of the Injection Molding Process			486	
15.6	Mappin	g with Dig	jimat	496	
15.7	Structur	ral Simula	tion with Ansys Workbench	497	
16	Sensor	Technolo	ogy for Injection Molding	507	
16.1	Simulat Schäfer	ion of the Box Mold	Cavity Pressure Curve Using the	510	
16.2	Simulat	ion of Cavi Now Spiral	ity Pressure and Specific Melt Pressure Based	512	
16.3	Process	Control by	y Means of Sensors	516	

17	Proces: Experin	s Optimization by Means of Statistical Design of nents (DoE)	519	
17.1	Robust	Design of the Injection Molding Process	520	
17.2	Use of M	Matrix Experiments to Characterize a Process	522	
17.3	Distinct Experin	ion between "Robust Design" and Statistical Design of nents	526	
17.4	Process	Optimization Using Autodesk Moldflow	532	
17.5	Process	Optimization Using Cadmould Varimos	540	
17.6	Selectio	n of the Quality Characteristics	543	
17.7	Analysi	s of the Statistical Experimental Design/Optimization	547	
17.8	Weighti	ng and Reduction in Quality Characteristics	552	
18	Special	Injection Molding Processes	555	
18.1	Sandwi	ch Injection Molding Basics	555	
18.2	Simulat	ion of Sandwich Injection Molding	559	
18.3	Fundan	nentals 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 $\ \ldots \ \ldots$	571	
	18.3.4	Simulation of Gas Injection Molding: Large-Area, Thick-Walled Molded Part	575	
19	Crossli	nking Molding Compounds	579	
19.1	General	Information on Crosslinking Molding Compounds	579	
19.2	Flow Be	havior of Crosslinking Molding Compounds	580	
19.3	Simulation Example for Liquid Silicone Rubber (LSR)			
	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-F	riendly Optimization of CAD Models with CATIA V5 \ldots	599
20.1	Evaluation of the Input Data		
20.2	Guidelines for Optimizing the Data		
20.3	Analysi	s 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-U	p 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	Simplify	ying 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 6			
	20.6.1 stp Format	70		
	20.6.2 stl Format	73		
20.7	Removing the Optimization History	73		
20.8	Joining Bodies Using Boolean Operators	76		
20.9	Collision Analysis as a Means of Control	76		
20.10	Intersection of Areas as a Means of Control	77		
20.11	Coupling Hot Runner Nozzles to the Cavity	77		
20.12	Positioning and Scaling of Core and Molded Part	80		
Index		33		

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 manu-



facturer, 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 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.

Basics of Injection Molding

1.1 Components of an Injection Molding Machine

Almost one third of all plastics are processed by injection molding. This mastermolding 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):

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

Where:

$$\begin{split} t_{Z} &= \text{Cycle time} \\ t_{E} &= \text{Injection time} \\ t_{N} &= \text{Holding pressure time} \\ t_{RK} &= \text{Residual cooling time} \\ t_{Ejector/Handling} &= \text{Ejection time} \\ t_{WZ \text{ open/close}} &= \text{Mold opening and closing} \end{split}$$

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 ejector 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 hold-ing 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 semicrystalline 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 pvT data and the specific heat capacity. The 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 timeconsuming 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

- 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

- 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

- 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)

- 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 byproducts 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

- 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

- 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 form 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

- 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

- 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

- 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

- 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 turnarounds, 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

- 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

- 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

- 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 multiphase 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

- 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 socalled stress cracks, crazes form in the material (Figure 1.25). Crazes are cracklike, 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

- 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.





- 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

- 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

В

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

С

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 dwell 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

Н

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

Κ

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

Μ

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

Ν

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

0

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

Ρ

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

α

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

۷

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

Υ

Young's modulus 98, 501

Ζ

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