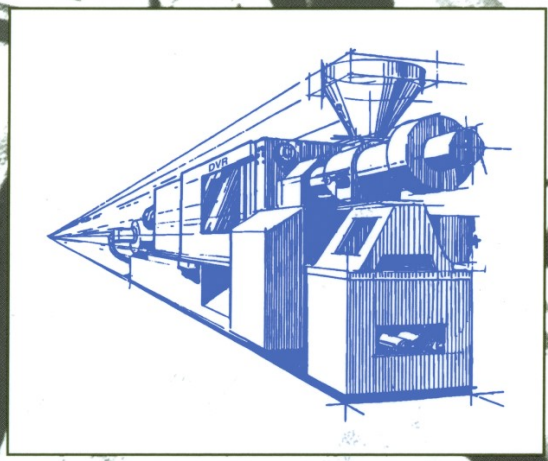


INJECTION MOLDING HANDBOOK

3rd Edition

Dominick V. Rosato
Donald V. Rosato
Marlene G. Rosato

Volume 1

Springer Science+Business Media, LLC

DNA Genotek Inc. Exhibit 2029 Page 1
Spectrum Solutions LLC v. DNA Genotek Inc. IPR2023-01424

INJECTION MOLDING HANDBOOK

INJECTION MOLDING HANDBOOK

THIRD EDITION

EDITED BY

DOMINICK V. ROSATO, P. E.

DONALD V. ROSATO, PH.D.

MARLENE G. ROSATO, P. E.



Springer Science+Business Media, LLC

DNA Genotek Inc. Exhibit 2029 Page 3
Spectrum Solutions LLC v. DNA Genotek Inc. IPR2023-01424

Library of Congress Cataloging-in-Publication Data

Injection molding handbook / Dominick V. Rosato, Donald V. Rosato, Marlene G. Rosato. – 3rd ed.
p. cm.
ISBN 978-1-4613-7077-2 ISBN 978-1-4615-4597-2 (eBook)
DOI 10.1007/978-1-4615-4597-2
1. Injection molding of plastics–Handbooks, manuals, etc. I. Rosato, Dominick V. II. Rosato, Donald V. III. Rosato, Marlene G.

TP1150.I55 2000
668.4'12—dc21

99-049946

Copyright © 2000 by Springer Science+Business Media New York
Originally published by Kluwer Academic Publishers in 2000
Softcover reprint of the hardcover 3rd edition 2000
All rights reserved. No part of this publication may be reproduced, stored in a retrieval system or transmitted in any form or by any means, mechanical, photo-copying, recording, or otherwise, without the prior written permission of the publisher, Springer Science+Business Media, LLC.

Printed on acid-free paper.

Contents

Preface **xxix**

Chapter 1 The Complete Injection Molding Process **1**

Introduction 1
Machine Characteristics 4
Molding Plastics 4
Molding Basics and Overview 4
People and Productivity 6; Plastic Materials 6; Morphology and Performance 9; Melt Flow and Rheology 11; Plasticating 12; Screw Designs 14; Molds 15; Processing 16; Process Controls 18; Control Guides 20; Art of Processing 21; Fine Tuning 21
Molding Operations 22
Automatic 22; Semiautomatic 22; Manual 22; Primary 23; Secondary 23
Purchasing and Handling Plastics 23
Processors 23
Captive 23; Custom 24; Proprietary 24
Training Programs 24
Processor Certifications 24
Plastics Machinery Industry 26
Summary 26

Chapter 2 Injection Molding Machines **28**

Introduction 28
Reciprocating (Single-Stage) Screw Machines 29
Two-Stage Machines 32
Injection Hydraulic Accumulator 32
Reciprocating vs. Two-Stage Machines 33
Other Machine Types 37
Machine Operating Systems 37
Hydraulic Operations 37
Reservoirs 40; Hydraulic Controls 42; Proportional Valves 42; Servovalves 43; Digital Hydraulic Control 43; Hydraulic Fluids and Influence of

DNA Genotek Inc. Exhibit 2029 Page 5
Spectrum Solutions LLC v. DNA Genotek Inc. IPR2023-01424

Heat 44; Pumps 44; Directional Valves 45; Servo and Proportional Valves 46
Electrical Operation 46
Electric Motors 47; Adjustable-Speed Drive Motors 47; Servo Drives 47; Microtechnology Moldings 47; Injection Molding: A Technology in Transition to Electrical Power 48
Hybrid Operations 58
Clamping Systems 59
Clamping Pressures 60; Hydraulic Clamps 61; Toggle Clamps 62; Hydromechanical Clamps 62; Hydroelectric Clamps 63; Comparison of Clamp Designs 64; Tie-bars 64; Tie-barless Systems 69; Platen Systems 71
Barrels 72
Barrel Borescoping 72; Barrel and Feed Unit 72; Barrel Heaters 73; Barrel Cooling 74; Barrel Characteristics 75
Screw Operations 75
Machine Sizes and Design Variations 75
Rebuilding and Repairs 79
Stripping, Polishing, and Plating 79; Machine Downsizing and Upsizing 79
Safety 80
Machine Lockout 80; Machine Safety 81; Identification of Hazards 82; Safety Built into the Machines 82; Current and Former Installations 88; IMM Safety Checklist 88; Safety Rules for Molding Department 88; American National Standard 92; Safety Standards 92; Plasticator Safety 93; Barrel-Cover Safety 93; Plant Safety 93; Safety Information 93
Designing Facilities 93
Upgrading 93; Clean Room 94; Clean Machines 94
Noise Generation 97
Startup and Shutdown Operations 98
Molding Operation Training Program 98
First Stage: Running an IMM 99; The Sequence in a Cycle 102; Second Stage: Parameter Setting and Starting a Job 105
Shear-Rate-Sensitive and -Insensitive Materials 109
Factors to Consider 113; Operating the Machine 127; Final Stage: Optimizing Molding Production 128; Specification Information, General 130; Specification Information, Details 131; Productivity and People 134; Training Information 136
Molding Guide 136
Guide to IMM Selection 137
Terminology 139

DNA Genotek Inc. Exhibit 2029 Page 6
Spectrum Solutions LLC v. DNA Genotek Inc. IPR2023-01424

Chapter 3 Plasticizing **151**
Introduction 151
Plasticators 151
Plastics Melt Flow 154; Barrel Temperature Override 157
Screw Sections 157
Feed Section 157; Transition Section 161; Metering Section 162
Elements of the Plasticating Processes 163
Screw Rotation 163; Soak Phenomena 164; Injection Stroke 165; Injection Pressure Required 166
Screw Plasticizing 168
Screw Design Basics 170; Sequence of Operations 172; Advantages of Screw Plasticizing 173; Length-to-Diameter Ratios 173; Compression Ratios 174; Rotation Speeds 175
Processing Thermoplastics or Thermoset Plastics 175
Screw Actions 176
Mechanical Requirements 177; Torque 177; Torque vs. Speed 177
Injection Rates 177
Back Pressures 178
Melt Performance 179
Melt Pumping 179
Melt Temperature 179
Temperature Sensitivity 179; Temperature Controls Required 179; Barrel Heating 180; Cooling 180
Melt Performance 181
Residence Time 181
Melt Cushions 181
Melt Shear Rate 181
Melt Displacement Rate 181
Shot Size 181
Recovery Rate 182
Screw–Barrel Bridging 182
Vented Barrels 182
Overview 182; Basic Operations 184; Barrel-Venting Safety 188
Screw Designs 188
Design Basics 189; Design Performance 189; Mixing and Melting Devices 189; Screw Barriers 193; Specialized Screw Designs 196; Screw Tips 197; Influence of Screw Processing Plastics 201; Melt Quality 202; Materials of Construction 204
Screw Outputs 204
Influence of Screw and Barrel Wear on Output 204
Influence of the Material on Wear 205; Screw Wear 205; Production Variations 205; Screw Wear Inspections 207; Output Loss Due to Screw Wear

207; Screw Replacement 207; Screw Wear Protection 208
Purging 208
Patents Influence Screw Designs 210
Terminology 210

Chapter 4 Molds to Products 221
Overview 221
Interrelation of Plastic, Process, and Product 221; Molding Process Windows 221; Cycle Times 223; Molding Pressure Required 224; Products 224
Processing Plastics 224
Basics of Melt Flow 225; Mold Filling Hesitation 225; Melt Cushioning 225; Mold Filling Monitoring 225; Sink Marks 226
Mold Descriptions 226
Mold Basics 230
Mold Optimization 234
Computer Systems 235
Mold Types 236
Molds For Thermosets 238; Mold Classifications 241
Plastic Melt Behaviors 241
Cold-Slug Well 243; Melt Orientation 244;
Cavity Melt Flow 249
Fill Rates 250; Melt Temperature 250; Mold Temperature 250; Packing Pressure 251; Mold Geometry 251; Flash Guide 251
Molding Variables vs. Performance 252
Shot-To-Shot Variation 253
Cavities 254
Cavity Melt Flow Analyses 254; Cavity Melt Fountain Flow 254
Cavity Evaluation 255
Machine Size 258; Plasticizing Capacity 258; Economics 258; Cavity Draft 259; Cavity Packing 259; Cavity Surface 259
Clamping Forces 260
Contact Area at Parting Line 262
Sprue–Runner–Gate Systems 262
Sprues 263; Runner Systems 264; Gates 277; Gate Summary 287
Correcting Mold Filling Imbalances in Geometrically Balanced Runner Systems 289
Isolating Mold Variations in Multicavity Molds 291
Mold Components 292
Ejector Systems 293; Ejector Pin Strength 296; Sprue Pullers 300; Side Actions 300; Angle Pins 301; Cam Blocks 302; Stripper-Plate Ejection 302;

External-Positive-Return Systems 302; Cam Actuation 303; Sprue Bushing and Locating Ring 303; Ring and Bar Ejection 303; Top-and-Bottom Ejection 304; Inserts 305; Side Guide Slides 307; Ejector Blades 307

Mold Venting 307

Molds for Thermoset Plastics 313

Mold Construction 313; Cold-Runner Systems 314; Injection–Compression Moldings 314

Mold Cooling 314

Overview 314; Design Considerations 315; Basic Principles of Heat Flow 317; Heat Transfer by Heat Pipes 321; Heat Balance of Halves 321; Mold Connection for Fluid 321; Cooling Time 321; Cooling with Melt Pulses 322; Flood Cooling 322; Spiral Cooling 322; Cooling Rates 322; Cooling Temperatures 322; Cooling Flow Meters 323

Undercuts 323

Mold Shrinkages and Tolerances 325

Shrinkage vs. Cycle Time 329

Ejection of Molded Products 332

Mold Release Agents 334

Mold Materials of Construction 334

Steels 334; Heat Treating 342; Requirements to be Met by Mold Steel 342; Aluminum 343; Beryllium–Copper 343; Kirksite 343; Brass 343

Etching Cavity Surfaces 344

Machining Safety 344

Moldmaker Directory 344

Mold Material Selection Software 344

Fabrication of Components 345

Hobbing 346; Cast Cavities 346; Electroforming 346; Electric-Discharge Machining 346

Tooling 347

Polishing 347

SPI Finish Numbers 348; Hand Benching 349; Direction of Benching 350; Ultrasonic Tools 351; Textured Cavities 351; Patterns of Different Textures 351; Mold Steels 352; Conditions Required for Polishing 352

Platings, Coatings, and Heat Treatments 353

Nickel 355; Chrome 355; Nitriding and Carburizing 356; Other Plating Treatments 357; Coating Treatments 357; Heat Treatments 358

Cleaning Molds and Machine Parts 359

Overview 359; Manual Cleaning 362; Oven Cleaning 362; Solvent Cleaning 362; Triethylene Glycol Cleaning 363; Postcleaning 363; Salt Bath Cleaning 363; Ultrasonic Solvent Cleaning 363; Fluidized-Bed Cleaning 363; Vacuum Pyrolysis Cleaning 363

Strength Requirements for Molds 364
Stress Level in Steel 364; Pillar Supports 365; Steel and Size of Mold Base 366
Deformation of Mold 367
Mold Filling 367; Deflection of Mold Side Walls 368
Eyebolt Holes 371
Quick Mold Change 371
Mold Protection 374
Automatic Systems 374; Heavy Molds 374
Preengineered Molds 378
Standardized Mold Base Assemblies 380
Specialty Mold Components 381
Collapsible and Expandable Core Molds 386
Prototyping 387
Overview 387; Stereolithography 387; Rapid Tooling 388
Buying Molds 389
Introduction 389; Industry Guide 389; Purchase Order 390; Mold Design 390; Production of Molds 392
Mold Storage 393
Computer-Aided Mold and Product Design 393
Production Control Systems 393
Computer Monitoring of Information 394
Productivity and People 394
Value Analyses 394
Zero Defects 395
Terminology 395

Chapter 5 Fundamentals of Designing Products 415
Overview 415
Molding Influences Product Performance 417
Design Optimization 421
Computer Analysis 422
Material Optimization 423
Material Characteristics 423
Behavior of Plastics 431
Thermal Stresses 437; Viscoelastic Behavior 437
Molding Tolerances 439
Tolerances and Designs 443; Tolerance Allowances 443; Tolerances and Shrinkages 444; Tolerances and Warpages 444; Thin-Wall Tolerances 444; Micron Tolerances 444; Tolerance Damage 444; Full Indicator Movements (FIMs) 444; Tolerance Selection 444; Tolerance Stack-Ups 445; Standard Tolerances 445
Tolerance Measurement and Quenching 447
Dimensional Properties 448
Dimensional Tolerances 449
Product Specifications 449; Using Geometric Tolerancing 450

Design Features That Influence Performance 451
Plastics Memory 451
Residence Time 453
Computerized Knowledge-Based Engineering 453
Orientation 453
Accidental Orientation 453; Orientation and Chemical Properties 453; Orientation and Mechanical Properties 454; Orientation and Optical Properties 454; Orientation Processing Characteristics 454; Orientation and Cost 454
Molecular Orientation: Design of Integral Hinges 455
Interrelation of Material and Process with Design 455
Design Shapes 455
Shapes and Stiffness 456
Stress Relaxation 457
Predicting Performance 458
Choosing Materials and Design 458
Design Concept 458; Engineering Considerations 458
Design Considerations 459
Design Parameters 460; Types of Plastics 460
Long-Term Behavior of Plastics: Creep 461
Designing with Creep Data 463; Allowable Working Stress 465; Creep Behavior Guidelines 466
Design Examples 466
Stapler 466; Snap-Fits 467; Springs 467
Design Approach Example 467
Design Accuracy 467
Risks and the Products 472
Acceptable Risks 472; Acceptable Goals 473; Acceptable Packaging Risks 473; Risk Assessments 473; Fire Risks 473; Risk Management 473; Risk Retention 473
Perfection 474
Cost Modeling 474
Innovative Designs 474
Protect Designs 474
Summary 475
Molders' Contributions 476
Terminology 477

Chapter 6 Molding Materials 479
Overview 479
Definition of Plastics 484; Heat Profiles 488; Costs 489; Behavior of Plastics 490; Checking Materials Received 491
Neat Plastics 491
Polymer Synthesis and Compositions 491
Polymerization 493
Copolymers 493
Interpenetrating Networks 497
Graftings 498

Reactive Polymers 498
Compounds 498
Additives 501; Fillers 502; Reinforcements 502; Summary 502
Alloys and Blends 507
Thermoplastic and Thermoset Plastics 510
Thermoplastics 511; Thermoset Plastics 511; Cross-Linking 512; Cross-Linking Thermoplastics 512; Thermoplastic Vulcanizates (TPVs) 512; Curing 512; Heat Profiles 513
Liquid Crystal Plastics (LCPs) 513
Elastomers, Thermoplastic, and Thermoset 514
Thermoplastic Elastomers 515; Thermoset Elastomers 515; Natural Rubbers 515; Rubber Elasticity 515; Rubber Market 515
Commodity and Engineering Plastics 515
Injection Molding Thermoplastics and Thermosets 516
High Performance Reinforced Moldings 516
Injection Moldings 518; Bulk Molding Compounds (BMCs) 518; Characterizations 519; Directional Properties 521
Viscosities 521
Newtonian Flow 522; Non-Newtonian Flow 523
Viscoelasticities 523
Plastic Structures and Morphology 523
Chemical and Physical Characteristics 524; Crystalline and Amorphous Plastics 524; Catalysts and Metallocenes 526; Plastic Green Strength 527
Molecular Weight (*MW*) 527
Average Molecular Weight 527; Molecular Weight Distribution 529; Additives 529; Molecular Weight and Melt Flow 530; Molecular Weight and Aging 530
Rheology and Melt Flow 530
Flow 531; Viscosity 531; Viscoelasticity 532; Intrinsic Viscosity 533; Shear Rate 533; Laminar and Nonlaminar Melt Flows 535; Melt Flow Analyses 535; Melt Flow Analysis Programs 535; Analyzing Melt Flow Results 536; Melt Flow Defects 536; Hindering Melt Flow with Additives 536; Melt Fractures 536
Cavity Filling 536
Plastic Raw Materials 537
Plastic Advantages and Disadvantages 537
Plastic Properties and Characteristics 537
Melt Shear Behaviors 537
Weld Line Strengths and Materials 541
Material Selections 548
Colorants 548; Concentrates 549; Barrier Plastics 549
ASTM 4000 Standard Guide for Plastic Classifications 550

DNA Genotek Inc. Exhibit 2029 Page 12
Spectrum Solutions LLC v. DNA Genotek Inc. IPR2023-01424

Thermal Properties and Processability 554
Melt Temperatures 554; Glass Transition Temperatures 555; Dimensional Stabilities 555; Thermal Conductivities and Thermal Insulation 556; Heat Capacities 556; Thermal Diffusivities 556; Coefficients of Thermal Expansion 556; Thermal Stresses 556
Shrinkages 556
Drying 557
Material Handling 557
Annealing 558
Recycling 558
Recycled Plastic Definitions 559; Recycled Plastic Identified 560; Recycled Plastic Properties 560; Recycling Size Reductions 560; Recycling Mixed Plastics 560; Integrated Recycling 560; Recycling Methods and Economic Evaluations 560; Recycling and Lifecycle Analysis 561; Recycling Commingled Plastics 561; Recycling Automatically Sorting Plastics 561; Recycling and Common Sense 561; Recycling Limitations 561
Recycling Facts and Myths 561
Warehousing 562
Storage and Condensation 562; Material Storage 562; Silo Storage 562
Processing Different Plastics 563
Polyethylenes 563
Molding Conditions 564; Materials 565; Molding Test Results 565
Polypropylenes 568
Molding Conditions 570
Copolyesters 573
Molding Conditions 573; Purging 574; Shutdown and Start-Up 574; Thermal and Rheological Properties 574; Drying 574; Mechanical Properties 575; Chemical Resistance 575; Weatherability 575; Color 575
Polyvinyl Chloride 575
Formulations 576; Molding Conditions 576; Screw Design 577; Material Handling Equipment 578; Processing Parameters 579; Problem Solving 579; Splay 579
Nylons 579
Molding Conditions 581; Performance Parameters 585; Design Parameters 586; Molding Performance Parameters 591; Mold Release 593; Close Tolerance: Fast Cycles 595; Recycling Plastics 596
ABSs (Acrylonitrile-Butadiene-Styrenes) 597
Molding Variables and Cause-and-Effect Links 597; Molding Variables and Property Responses 599; Appearance Properties 599; Warping 600;

Mechanical Properties and Molding Variables 601; Izod impact 602; Molding for Electroplating 605; Property Variation with Position Mold Geometry 605; Summary 606
Polycarbonates 606
Drying 606; Recycle and Virgin Proportions 607; Processing 608; Hydrolysis 609; Rheology 609; Heat Transfer 609; Residual Stress 610; Annealing 611
Injection Molding Thermosets 611
Process 613; Hot- and Cold-Runner Molding 614; Material Stuffer 615
Energy Considerations 616
Summary 617
Terminology 617

Chapter 7 Process Control 623

Process Control Basics 623
Developing Melt and Flow Control 630; Inspection 630; Computer Process Data Acquisition 630; Control Flow Diagrams 632; Fishbone Diagram 632
Overview 634
Technology 636; Fast Response Controls 638; Control Approaches 639; Process Control Methods 640; Production Monitoring 640; On-Machine Monitoring 641
Temperature Control of Barrel and Melt 644
Electronic Controls 646
Fuzzy Logic Control 647
Process Control Techniques 648
Process Control Approaches 652
What Are the Variables? 652; Why Have Process Control? 654; Control of Which Parameters Can Best Eliminate Variability? 654; What Enables Parameter Controllability? 657; Where Does the Process Controller Go? 661; Basic Features a Process Controller Should Have 662; Applications 664; Summary 666
Process Control Problems 667
Cavity Melt Flow Analyses 668
Problem 669; Melt Viscosities versus Fill and Pack 669; Test Methodology 670; Analyzing Results 673; Example Test 673; Using Empirical Test Data to Optimize Fill Rates 674; Melt Vibrations during Filling 675; Stabilizing via Screw Return Time 675
Relating Process Control to Product Performances 676
Sensor Requirements 676; Molding Parameters 676; Display of Monitored Molding Parameters

678; Machine Controls 678; Microprocessor Advantages 679
Types of Instruments 680
Functions 680; Rotary and Linear Motion 680
Adaptive Control: *PVT* and *PMT* Concepts 681
Optimization via PVT 681; PMT Concept 683
Controllers 684
Designs 684
Sensor Control Responses 685
Transducers 685
Linear Displacement Transducers 685; Linear Velocity Displacement Transducers 686; Pressure Transducers 686; Transducer Calibrations 686; Transducer Environments 686
Transputer Controllers 686
Temperature Controllers 687
Temperature Variations 688; Melt Temperature Profiles 690; Automatic Tuning 691; Temperature Sensors 691; Fuzzy Logic Controls 692; Fuzzy-PID Controls 692
Temperature Timing and Sequencing 692
Pressure Controls 692
Screw Tips 692; Cavity Fillings 692
Pressure PID Controls 693
PID Tuning: What It Means 693; The Need for Rate Control on High-Speed Machines 694
Fuzzy-Pressure Controls 694
Injection Molding Holding Pressures 695
Process Control Fill and Pack 695
Process Control Parameter Variables 695
Adaptive Ram Programmers 696
Injection Molding Boost Cutoff or Two-Stage Control 697
Injection Molding Controller Three-Stage Systems 701
Three-Stage Systems 701
Mold Cavity Pressure Variables 702
Programmed Molding 702
Parting Line Controls 702; Computer Microprocessor Controls 703; Computer Processing Control Automation 703
Molding Thin Walls 703
Control System Reliabilities 703
Operations Optimized 704
Control Tradeoffs 704
Process Control Limitations and Troubleshooting 704
Control 705; Tie-Bar Growth 706; Tie-Bar Elongation 706; Thermal Mold Growth 706; Shot-to-Shot Variation 706
Intelligent Processing 709
Intelligent Communications 709; Systematic Intelligent Processing 710

Processing Rules 710
Processing and Patience 710
Processing Improvements 710
Control Advantages 711
Plantwide Control and Management 711
Automatic Detections 712
Terminology 713

Chapter 8 Design Features That Influence Product Performance 716
Overview 716
Audits 717; Computer Approaches 717; Design Feature That Influence Performance 718
Plastic Product Failures 718
Design Failure Theory 719
Basic Detractors and Constraints 719
Tolerance and Shrinkage 721; Residual Stress 725; Stress Concentration 726; Sink Mark 727
Design Concept 727
Terminology 730
Sharp Corners 730
Uniform Wall Thickness 732
Wall Thickness Tolerance 732
Flow Pattern 733
Parting Lines 733
Gate Size and Location 733
Taper or Draft Angle 735
Weld Lines 738
Meld Lines 740
Vent, Trapped Air, and Ejector 740
Undercuts 740
Blind Holes 740
Bosses 747
Coring 750
Press Fits 751
Internal Plastic Threads 752
External Plastic Threads 752
Molded-In Inserts 753
Screws for Mechanical Assembly 754
Gears 759
Ribs 760
Geometric Structural Reinforcement 763
Snap Joints 764
Integral Hinges 765
Mold Action 766

Chapter 9 Computer Operations 770
Overview 770
Communication Benefits 773; Computerized Databases of Plastics 775; CAD/CAM/CAE Methods 775; Computer-Integrated Manufacturing 775

Benefits of CAD/CAM/CAE for Mold Design 776
Productivity 776; Quality 777; Turnaround Time 778; Resource Utilization 778
Basics in CAD/CAM/CAE Modeling 778
Mechanical Design 779; Computer-Aided Engineering 780
Mold Flow Analysis 781
Product Designers 783; Mold Designers and Moldmakers 784; Injection Molders 785
Basic Melt Flow Analysis 786
Multisections 789; Finite Element Techniques 790; Shrinkage and Warpage 791; Benefit Appraisal 795; Moldflow Basic Technology 795
Mold Cooling 796
Introduction 796; Fundamentals 799; Mold Cool Analysis 801
Modeling Methods Applied to Part and Mold Design 823
Wire Frame Modeling 824; Surface Modeling 826; Solids Modeling 828
Computer Capabilities for Part and Mold Design 829
Group Technology 829; Finite Element Modeling 830; Digitizing 831; Layering 832; Groups 833; Patterns 833; Large-Scale Geometry Manipulation 833; Local Coordinates or Construction Planes 834; Model and Drawing Modes and Associativity 834; Verification of Geometric Relationships 835; Automatic Dimensioning and Automatic Tolerance Analysis 836; Online Calculation Capabilities and Electronic Storage Areas 836
Illustration of Mold Design Process 836
The Manual (Paper) Method 837
The CAD/CAM/CAE Method 840
Online Databases 843
The Database Concept 843; Graphics Databases 844; Defining the Library Database 845
Tolerances and Dimensional Controls 846
Computer Controllers 846
CAD/CAM/CAE and CIM 847
Numerical Control Process 849
Programmable Controller Safety Devices 849
Computer Optical Data Storage 850
Artificial Intelligence 850
Computers and People 850
Computer-Based Training 850
Myths and Facts 850
Capability and Training 851
Computer Software 852
Molding Simulation Programs 854
RAPRA Free Internet Search Engine 854
Software and Database Programs 854

Injection Moldings and Molds 856; Materials 857; Shrinkage 858; Materials and Designs 859; Design Products 860; Engineering 861; Graphics 861; Management 862; General Information 862; Training 862
Plastics, Toys, and Computer Limitations 863
Computers Not Designed for Home 863
Summary 863
Terminology 864

Chapter 10 Auxiliary Equipment and Secondary Operations 868
Introduction 868
Energy Conservation 870; Planning Ahead, Support Systems 871
Overview 871
Hoppers 871; Material Handling, Feeding, and Blending 872; Material Handling Methods 872; Sensors 874
Materials Handling 875
Bulk Density 875; Basic Principles of Pneumatic Conveying 876; Air Movers 883; Pneumatic Venturi Conveying 886; Powder Pumps 886; Piping 888; Hoppers 889; Filters 889; Bulk Storage 891; Blenders 891; Unloading Railcars and Tank Trucks 894
Drying Plastics 895
Nonhygroscopic Plastics 895; Hygroscopic Plastics 895; Drying Overview 895; Dryers 896
Water Chilling and Recovery 904
Overview 904; Heat-Transfer Calculations 905; Requirements Vary with Materials 905; Water Recovery 907; General Considerations 908; Calculation of the Cooling Load 911; Determining Water Loads 913
Energy-Saving Heat Pump Chillers 915
Granulators 916
Safety 916; Basics 917; Hoppers 917; Cutting Chambers 918; Cutting Chamber Assembly 921; Hard Face Welding 921; Screen Chambers 922; Auger Granulators 922; Granulating and Performance 924
Mold Dehumidification 929
Dewpoints 929; Mold Surface Temperatures 929; Effect of Change in Air Properties 930; Air Conditioning and Desiccant Dehumidification 931; Dehumidification System 932
Parts-Handling Equipment 933
Controlled Motions 933; People and PHE 935; Different Types 935; Value in Use 937; Detriments 938; Robots Performance 938; Safety Measures 938

Machining 939
Overview 939; Plastic Characteristics 939; Cutting Guidelines 940
Joining and Assembling 941
Adhesives 941; Solvents 946; Welding Techniques 948; Welding Process Economic Guide 953
Cleaning Tools 953
Abrasives 953; Carbon Dioxide 953; Cryogenic Deflashing 954; Brass 954; Hot Salts 954; Solvents 954; Ultrasonics 954; Vacuum Pyrolysis 954; Coatings 955
Finishing and Decorating 955
Potential Preparation Problems 955; Pretreatments 959; Removing Mold Release Residues 959
Terminology 963
Robot Terms 966

Chapter 11 Troubleshooting and Maintenance 969
Troubleshooting Introduction 969
Plastic Material and Equipment Variables 970
Definitions 971
Defects 972
Remote Controls 972
Troubleshooting Approaches 972
Finding the Fault 976
Shrinkages and Warpages 978
Weld Lines 978
Counterflow 979
Troubleshooting Guides 979
Flashes 980
Injection Structural Foams 994
Hot-Runners 994
Hot-Stamp Decorating 994
Paint-Lines 994
Granulator Rotors 1001
Auxiliary Equipment 1001
Screw Wear Guide 1001
Inspection Rollers 1010; Diameters 1010; Depths 1011; Concentricity and Straightness 1011; Hardness 1011; Finish and Coating Thickness 1012; Screw Manufacturing Tolerances 1012
Barrel Inspection Guide 1012
Inside Diameters 1012; Straightness and Concentricity 1012; Barrel Hardness 1012; Barrel Specifications 1012
Preventive Maintenance 1013
Cleaning the Plasticator Screw 1014; Oil Changes and Oil Leaks 1015; Checking Band Heaters, Thermocouples, and Instruments 1015; Alignment, Level, and Parallelism 1015; Hydraulic,

Pneumatic, and Cooling-Water Systems 1015; Hydraulic Hose 1016
Keep the Shop Clean 1016
Keep Spare Parts in Stock 1016
Return on Investment 1016
Maintenance 1018
Hydraulic Fluid Maintenance Procedures 1020; Problems and Solutions 1020; Downtime Maintenance 1021; Preventative Maintenance 1021; Services 1022
Safety 1023
Maintenance Software 1023
Summary 1023
Terminology 1023

Chapter 12 Testing, Inspection, and Quality Control 1028
Testing 1028
Design and Quality 1031
Basic versus Complex Tests 1031
Sampling 1032
Acceptable Quality Level 1032; Sampling Plan 1032; Sampling Size 1033
Characterizing Properties and Tests 1033
Orientation and Weld Lines 1033; Density and Specific Gravity 1035; Morphology: Amorphous and Crystalline Plastics 1036; Molecular Structures 1037
Mechanical Properties 1041
Mechanical Test Equipment 1042; Tensile Test 1042; Deflection Temperature under Load 1045; Creep Data 1045
Electrical Tests 1046
Thermal Properties 1046
Chemical Properties 1046
Chromatographic and Thermal Tests 1049
Liquid Chromatography 1049; Gel Permeation Chromatography 1049; Gas Chromatography 1050; Ion Chromatography 1050; Thermoanalytical Method 1051; Thermogravimetric Analysis 1051; Differential Scanning Calorimetry 1052; Thermomechanical Analysis 1053; Dynamic Mechanical Analysis 1054; Infrared Spectroscopy 1054; X-Ray Spectroscopy 1055; Nuclear Magnetic Resonance Spectroscopy 1055; Atomic Absorption Spectroscopy 1055; Raman Spectroscopy 1055; Transmission Electron Microscopy 1056; Optical Emission Spectroscopy 1056; Summary of Characterizing Properties 1056
Types of Tests 1060
Selected ASTM Tests 1062; Viscoelastic Properties 1079; Rheology, Viscosity, and Flow 1080;

Online Viscoelastic Measurements for Plastics Melt Processes 1080
Optical Analysis via Microtoming 1081
Thermal Properties 1084
Useful Temperature Range 1084; Glass Transition and Melt Temperatures 1084; Thermal Conductivity 1086; Heat Capacity 1086; Coefficient of Linear Thermal Expansion 1086; Temperature Dependence of Mechanical Properties 1089; Diffusion and Transport Properties 1091; Permeability 1091; Migration 1092
Overview of Plastic Properties 1092
Melt Tests 1095
Melt Flow Tests 1095; Melt Index Test 1095; Melt Index Fractional Tests 1098; Molding Index Tests 1098; Measurements 1098
Temperature Scales 1099
Types of Scales 1099
Nondestructive Tests 1099
Radiography 1099; Ultrasonics 1100; Liquid Penetrants 1100; Acoustics 1100; Photoelastic Stress Analysis 1100; Infrared Systems 1101; Vision System Inspections 1101; Computer Image Processors 1102
Computer Testing 1103
Drying Hygroscopic Plastics 1103
Determining Moisture Content 1103
Laboratory Organizations Worldwide 1104
American Society for Testing and Materials 1105; International Organization for Standardization 1105; Underwriters' Laboratory Classifications 1106
International System of Units 1106
Inspections 1106
Identification of Plastics 1107
Estimating Plastic Lifetimes 1107
Quality Control 1109
Quality Control Defined 1110; Quality Control Variables 1110
QC Begins When Plastics Are Received 1111
No More ABCs 1112; Need for Dependability 1112; Quality Auditing 1112
Reliability and Quality Control 1113
Failure Analysis 1113
Quality Control Methods 1113
Image Quality Indicators 1114
Quality Control and Quality Assurances 1114
Auditing by Variables Analysis 1115
Acceptable Quality Levels 1116
Quality Optimization Goals 1116
Quality System Regulation 1117

DNA Genotek Inc. Exhibit 2029 Page 21
Spectrum Solutions LLC v. DNA Genotek Inc. IPR2023-01424

Total Quality Management 1117
Training and People 1117
Training and Quality 1117
Emerging Trends in Training 1117
Training versus Education 1118
Economic Significance of Quality 1118
Cost of Quality 1119
Terminology 1119

Chapter 13 Statistical Process Control and Quality Control 1127
Overview 1127
Combining Online SPC and Offline SQC 1127; Improve Quality and Increase Profits 1128; Statistical Material Selections: Reliabilities 1128; Statistical Material Selections: Uncertainties That Are Nonstatistical 1129; Statistical Probabilities and Quality Control 1129; Statistics and Commitments 1129; Statistics and Injection Molding 1129 Computers and Statistics 1131; Statistical Tools 1134
Online Monitoring of Process Variables 1134
Gathering and Analyzing Data 1135
Process Control and Process Capability 1138
Control Charts 1138
Defect Prevention 1139
Understanding Modern Methods of Control 1140
Standard Deviations 1142; Frequency Distribution 1143; Control Chart 1145
Standard Deviation versus Range 1147
Basic Statistical Concepts 1148
Mean Value, Range, and Standard Deviation 1148; Distribution 1149; Process Control Chart 1150; Machine Capability 1150; Process Capability 1150
Importance of Control Charts 1151
Practical Example 1152
Machine Capability 1153; Process Capability 1153; Control Limits for the Process Control Chart 1154
A Successful SPC System 1154
Production Controls 1155; SPC Step One: Raw Material 1156; SPC Step Two: Materials Handling 1156; SPC Step Three: Injection Molding 1156; SPC Implementation: Summary of Experience 1156
How to Succeed with SPC 1159
Outlook 1160
Terminology 1160

Chapter 14 Costing, Economics, and Management 1163
Overview 1163
Machine Sales 1163; Formulas for Business Failures 1164; Managing 1164
Costing 1165
Estimating Part Cost 1167; Automation of Data Gathering 1169; Machinery Financing 1169; Energy Savings 1170
Technical Cost Modeling 1171
Cost Analysis Methods 1171
Material Times Two 1171; Material Cost plus Shop Time 1172; Material Cost plus Loaded Shop Time 1172; Quotes 1172
Technical Cost Analysis 1173
Variable Cost Elements 1173; Fixed Costs 1174; Summary of Fixed and Variable Costs 1177; Process Parameters 1178; Technical Cost Modeling 1178; Summary of Technical Cost Analysis 1179
Financial Plant Management 1180
Cost Management 1180
Information Necessary for Product Costing and Cost Control 1182; Reporting from the Production Floor and Management Control Reports 1183
Profit Planning and Budgeting 1185
Gathering the Data for Profit Planning and Budgeting 1186; Establishing Profit, Goals, and Sales Forecasts 1186; Developing the Detailed Plans and Budgets 1187; Flexible Budgeting 1187
Materials Management 1188
Order Processing 1188; Inventory Control 1189; Production Scheduling and Control 1189; Scheduling Approaches 1190; Purchasing 1191
Terminology 1192

Chapter 15 Specialized Injection Molding Processes 1197
Introduction 1197
Blow Moldings 1197
Injection Blow Moldings 1201; Stretched Blow Moldings 1204; Stretched Blow Moldings with Handle 1206; Stretched Blow Molding Operation Specialties 1207; Blow Molding Shrinkages 1209; Troubleshooting 1211; Blow Molding versus Injection Molding 1215
Coinjection Molding 1216
Injection Molding Sandwich Structures 1218
Gas-Assist Injection Molding 1219
Advantages and Disadvantages 1220; Basic Processes and Procedures 1220; Molding Aspects 1223; Shrinkage 1224; Summary 1224

Gas Counterflow Molding 1225
Melt Counterflow Molding 1225
Structural Foam Molding 1225
Overview 1225; Performance 1226; Plastic Materials 1226; Characteristics of Foam 1226; Design Analysis 1227; Blowing Agents 1229; Methods of Processing SF with Chemical Blowing Agents 1230; Processing SF with Gas Blowing Agents 1232; Tooling 1234; Start-up for Molding 1234
Injection–Compression Molding (Coining) 1235
Multiline Molding 1236
Counterflow Molding 1236
Oscillatory Molding of Optical Compact Disks 1237
Digital Video Disk Moldings 1238
Continuous Injection Molding 1239
Velcro Strips 1239; Electrically Insulated Buttons for Coaxial Cables 1242; Railtrack Molding 1243
Reaction Injection Molding 1244
The Mold 1248; Process Controls 1249
Liquid Injection Molding 1250
Soluble Core Molding 1251
Insert Molding 1252
Inmolding 1252
Two-Color Molding 1253; Decoration 1253; Paint Coating 1254; Back Molding 1254; Two-Shot Molding 1254; Inmold Assemblies 1254; Double-Daylight Process 1255
Overmolding Compatible Plastics with No Welding 1255
Closure Moldings 1256
Unscrewing Closures 1256; Conventional Unscrewing Molds 1256; Unscrewing System Moldings 1256; Collapsible and Expandable Core Molds 1257; Split-Cavity Molds 1258; Strippable Thread Molds 1258
Vacuum Molding 1260
Tandem Injection Molding 1260
Molding Melt Flow Oscillations 1261
Ram Injection Molding 1262
Golf Ball Moldings 1262
Micro Injection Molding 1264
Aircraft Canopies 1265
Injection Molding Nonplastics 1266
Introduction 1266; Metal Injection Molding 1266; Ceramic Injection Molding 1268
Terminology 1268

Chapter 16 Injection Molding Competition 1270
Introduction 1270
Plastic Fabricating Processes 1272
Rotational Molding 1274
Extrusions 1283

Extrusion Blow Moldings 1284
Formings 1288
Thermoforming 1289
Molds 1291
Cold Forming 1291
Cold Draw Forming 1292
Dip Forming 1292
Pressure Forming 1292
Rubber Pad Forming 1292
Compression-Stretched Moldings 1293
Solid-Phase Scrapless Forming 1293
Solid-Phase Pressure Forming 1293
Slip Forming 1293
Castings 1293
Foam Molding 1294
Expandable Plastics 1294
Expandable Polystyrenes 1294
Compression Molding 1295
Laminates 1297
Transfer Molding 1298
Screw Plunger Transfer Molding 1298
Reinforced Plastics 1298
Directional Properties 1301; Processes and Products 1301
Stampable Reinforced Plastics 1303
Machining Plastics 1304
Processor Competition 1304
Legal Matters 1304
Accident Reports 1304; Acknowledgments 1304; Chapter 11 Act 1304; Conflicts of Interest 1304; Consumer Product Safety Act 1304; Copyright 1305; Defendant 1305; Employee Invention Assignment 1305; Expert Witness 1305; Insurance Risk Retention Act 1305; Invention 1305; Mold Contractional Obligation 1305; Patent 1305; Patentability 1306; Patent Information 1306; Patent Infringement 1306; Patent Pooling with Competitors 1306; Patent Search 1306; Patent Term Extension 1306; Patent Terminology 1306; Plaintiff 1306; Processor, Contract 1307; Product Liability Law 1307; Protection Strategies 1307; Quotations 1307; Right-To-Know 1307; Shop-Right 1307; Software and Patents 1307; Tariff 1307; Term 1307; Tort Liability 1308; Trademark 1308; Trade Name 1308; Warranty 1308

Chapter 17 Summary 1309
The Most Important Forming Technique 1309
Processing Trends 1311
Productivity 1313
Machine Aging 1315; Response to Change 1316

Process and Material Selections 1318
Plastics and Equipment Consumption 1318
Machinery Sales 1318
Trends in Machinery 1318; Computers and Injection Molding 1320; Interfacing Machine Performance 1320
Molding in an Industrialized Country 1321
Compromises Must Frequently Be Made 1321
Standard Industrial Classification 1322
Plastic Industry Size 1322
Energy and Plastics 1323
Plastic Data: Theoretical Versus Actual Values 1324
Markets 1324
Packaging 1325; Velcro for Flexible Packaging 1327; Building and Construction 1327; Lumber 1327; Pallets 1327; Automotive Parts 1329; Printed Circuit Boards and Surface Mounted Technology 1330; U.S. Postal Service 1330; Medical Applications 1330; Toilets and Water Conservation 1330; Bearings 1330; Blow Molding Innovations 1330; Beer Bottles 1331; Collapsible Squeeze Tubes 1331; Asthma Inhalers 1331
Economic Control of Equipment 1331
Automated Production 1334; Energy Savings 1335
Management and People 1337
Discipline 1337; Productivity 1338; Experience 1338; Plant Controls 1338
Analysis of Plastics Affecting Business Strategies 1339
Example 1 1339; Example 2 1339; Example 3 1340
Correcting Misperceptions about Plastics 1341
Myths and Facts 1341; Limited Oil Resources 1342; Limited U.S. Steel Resources 1342; Plastic Advocates 1342
Solid Waste Problem and Product Design Solutions 1342
Statistics: Fact and Fiction 1344; Landfill 1345; Recycling 1345; Incineration 1345; Degradable 1346
Analyze Failures 1346
Creativity 1347
Innovations and the Markets 1348; Industrial Designers 1348; Da Vinci's Creativity 1348
Design Successes 1349
Target for Zero Defects 1349
Excess Information: So What's New? 1349
Fabricating Employment 1350
History 1350
Barrel History 1351; Hopper Magnet 1352; Blow Molding 1352; Coca-Cola Bottle 1353; Coor's Beer Bottle 1353; Recycling History 1353; Squeeze Tube 1353; Zipper 1353; Waste Containers 1354; Shotgun Shells 1354; Water Treatment 1354

Profits 1354
Profits and Time 1354
Plastics, Cradle-to-Grave 1355
Future for Injection Molded Plastics 1355
Injection Molding in the Forefront 1356
Summary 1356

Appendices
1. Abbreviations 1359
2. Conversions 1374
3. Symbols and Signs 1381
4. Web Sites on Plastics 1383

References 1395

About the Authors 1411

Index 1413

4

Molds to Products

Overview

In the manufacture of molded products there is always a challenge to utilize advanced techniques such as monitoring and control systems (Chap. 7), statistical analysis, and so on. However, these techniques are only helpful if the basic operations of molding are understood and characterized to ensure the elimination or significant reduction of potential problems. This understanding encompasses factors such as mold design and operation, plastic performance during melting (Table 4-1), and the operation of the injection molding machine (IMM) to produce cost–performance-effective products at a profit (Chaps. 8 and 14).

Interrelation of Plastic, Process, and Product

In order to fabricate a cost–performance-effective molded product and understand potential problems and their solutions, it is helpful to consider the relationships of machine and mold capabilities, plastics processing variables, and product performance. A distinction has to be made between machine conditions and processing variables. Machine conditions include operating temperature and pressure, mold and/or die temperature, machine output rate, etc. Processing variables are more specific, such as the melt condition in the mold, flow rate vs. temperature, etc.

Molding Process Windows

Process windows are the ranges of processing conditions, such as melt temperature, pressure, and shear rate, within which a specific plastic can be fabricated with acceptable or optimum properties by a particular fabricating process (1, 7, 515). A window is a defined "area" or "volume" in the space of a processing system's process control variables. The window for a specific plastic part can vary significantly if changes are made in its design and the fabricating equipment used (Chap. 8). Also important is the uniformity of the plastic material (pellets, flakes, etc.). The greater the uniformity, the easier it is to control the process and improve product quality (see the section on Plastic Material and Equipment Variables in Chap. 11).

By plotting injection pressure (ram pressure) vs. mold temperature, a molding area diagram (MAD) is developed that shows the best combinations of pressure and temperature to produce quality parts. The size of the diagram (Fig. 4-1) shows the molder's latitude in producing good parts. To mold parts at the lowest cycle time, the molding machine would be set at the lowest temperature and highest pressure location on this diagram.

DNA Genotek Inc. Exhibit 2029 Page 28
Spectrum Solutions LLC v. DNA Genotek Inc. IPR2023-01424

Table 4-1 Examples of plasticizing processing temperatures[a]

Polymer	Type	T_g, °F (°C)	Processing Temperature, °F (°C)
Polyetheretherketone (PEEK)	Semicrystalline	290 (143)	650 (343)
Polyphenylene sulfide (PPS)	Semicrystalline	185 (85)	630 (332)
Polyaryleneketone	Semicrystalline	400 (204)	700–780 (371–416)
Polyarylene sulfide	Amorphous	410 (210)	625–650 (329–343)
Polyetherimide (PEI)	Amorphous	Varies: 450 (232) to 545 (285)	Varies: 575–650 (302–343) to 650–700 (343–371)
Polyarylether	Amorphous	476 (247)	650 (343)
Polyethersulfone (PES)	Amorphous	510 (266)	575 (302)
Polyamide-imide (PAI)	Amorphous	470 (243)	650 (343)
Polyimide	Pseudothermoplastic	470 (249)	680 (360)
		482 (250)	660 (349)
		536 (280)	660 (349)
		536 (280)	660 (349)

[a] Typical commodity TPs use about 400 to 550°F (204 to 288°C).

If due to machine and plastic variables rejects develop, then one moves the machine controls to achieve higher temperatures and/or lower pressures and thus restore quality. This is a simplified approach to producing quality parts, since only two variables are being controlled. (This example uses a thermoplastic; with a thermoset, to reduce cycle time the highest temperature and pressure would be used, etc.)

The next step in the molding-area technique is to use a three-dimensional diagram (Fig. 4-2). By plotting melt temperature vs. injection pressure vs. mold temperature, one obtains a molding volume diagram (MVD), providing more precision control in setting the machine.

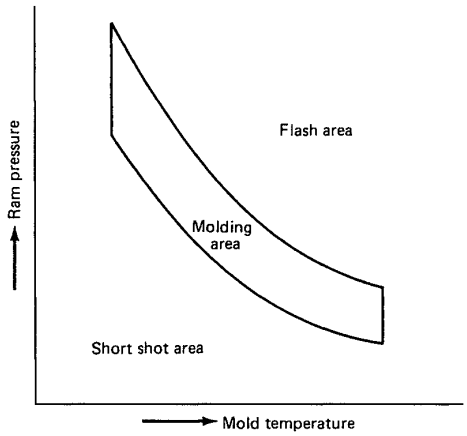


Fig. 4-1 Molding area diagram.

Developing the actual data involves slowly increasing the ram (injection) pressure until a value is obtained at which the mold is just filled out. This is referred to as the minimum fill pressure for that combination of material, mold temperature, and melt temperature. The ram pressure is then increased until the mold flashes. This is logged as the maximum flash pressure. These two pressure values then represent a set of data points for one combination of melt and mold temperatures.

Next, the melt temperature is changed (leaving the mold temperature constant), and a new set of minimum and maximum pressures determined. This is continued until the maximum and minimum melt temperatures are found.

Then the mold temperature is changed, and all the above repeated until the maximum and minimum mold temperatures are found. Once the data are obtained, three-dimensional MVDs are constructed.

MVDs show that the melt temperature for injection molding plastic is an important variable that was not evident in two-dimensional MADs. MVDs are used with all

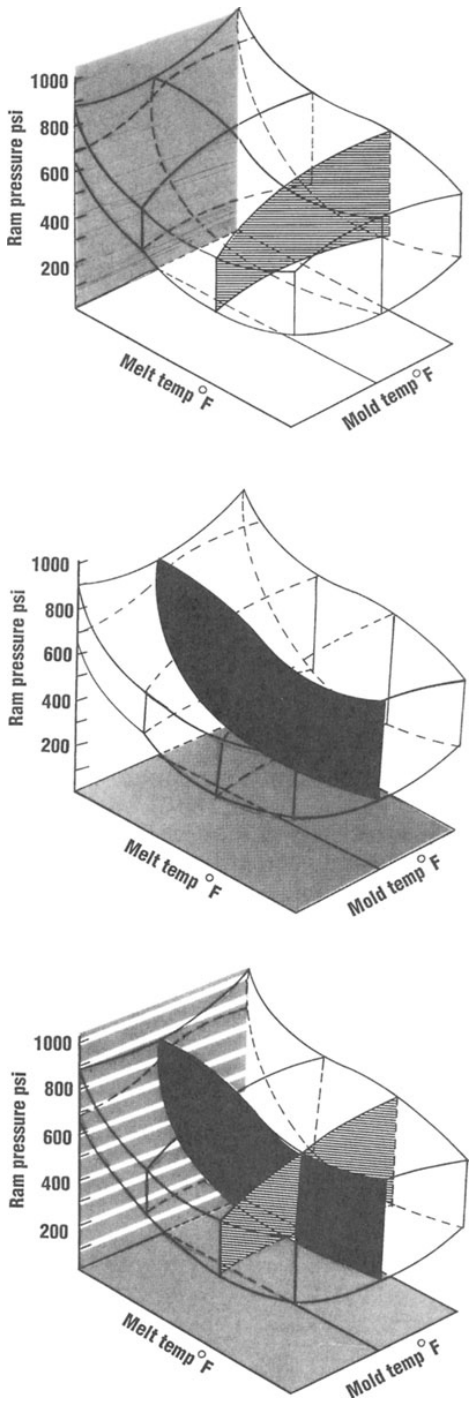


Fig. 4-2 Molding volume diagram showing three steps.

thermosets and thermoplastics. The significance of the MVD approach lies in the fact that one ends up with a dramatic and easily comprehended visual aid to analyzing three of the most important variables for injection molding—namely, injection pressure, mold (or barrel for thermoplastics) temperature, and melt temperature (1, 7, 283).

Using this two- and three-dimensional approach for making molding diagrams, you can analyze injection rate, cavity pressure, etc., and also consider whether to use manual or automatic process controls. As discussed in this chapter and Chap. 9, the use of automatic controls makes it easier to set controls and ensure quality. Of course, some molds produce quality parts just with manual controls; most of the 80,000 injection molding machines in the United States use only manual controls. However, major changes are occurring because the automatic controls can significantly reduce cost and provide zero (or practically zero) defects.

Cycle Times

A cycle is the complete repeating sequence of operations in a process or part of a process. One cycle time is the time period, or elapsed time, between a certain point in one cycle and the same point in the next cycle; it is the time to mold a part. As a general guide, regardless of the plastic processed, the average wall thickness (in thousandths of inches) multiplied by 250 equals the cycle time in seconds.

The problem of shortening the cycle time lies principally in assessing all the difficulties of the injection molding process during the design of the part and the mold. Thus what is needed is a device for achieving optimum designs of part and mold. Program systems that provide for computer simulation of the injection molding process are used for this purpose. One should keep abreast of the availability and performance of relevant software so that one can gain in experience. Most important are programs to reduce the cycle time by evaluating the actual process operational settings (see the section on Molding Simulation Programs in Chap. 9).

DNA Genotek Inc. Exhibit 2029 Page 30
Spectrum Solutions LLC v. DNA Genotek Inc. IPR2023-01424

Molding Pressure Required

The molding pressure is the pressure applied to the molding material in the mold cavity or cavities during injection of the melt from the plasticator. The pressure required is based on the projected area taken at right angles to the applied force (clamp closing direction) plus the cross-sectional areas of those runners that solidify on the mold parting line. The melt pressure required for a specific material is determined from past experience and/or from the material supplier's data sheets.

The force required is calculated by multiplying the projected area by the melt pressure. It is expressed in psi or MPa. The result is the total clamping force required (usually converted to tons). To ensure sufficient pressure in practice, consider multiplying by a processing safety factor (SF) of 1.1. With experience, however, this SF can be reduced or even eliminated (see the section on Molding Thin Walls in Chap. 7).

Products

Plastic products are used in all industries (Chap. 17). They can range from parts weighing an ounce (indeed, grams) to hundreds of pounds. Typical products are reviewed throughout this book. Figure 4-3 is an example of diverse molded and other products used in an electric pressing iron.

Shapes Both shape geometry and design are heavily process-related. As an example, the ability to mold ribs may depend on the thickness and length of the rib, the ability of the melt to flow adequately during processing, the flowability of a plastic reinforced with glass fiber or other reinforcements or fillers, and so on. The ability to produce hollow shapes may depend on the ability to use removable cores or inserts or techniques that include air, fusible, or soluble solids, and even sand (Chap. 15). Hollow parts can be produced using cores that remain in the part, such as foam inserts.

Product obsolescence Tradeoffs exist between coming up with a new design and providing incremental improvements to an existing product. A new design usually avoids constraints imposed by the incremental approach, but it can be costly in time and resources. If one continues the incremental approach too long, the entire concept runs the risk of becoming obsolete.

Processing Plastics

Injection molding machinery provides the capability to process different plastics that require different methods of operation. There are other specifications for IMMs than meeting product size requirements. Machines must also be designed to meet the process

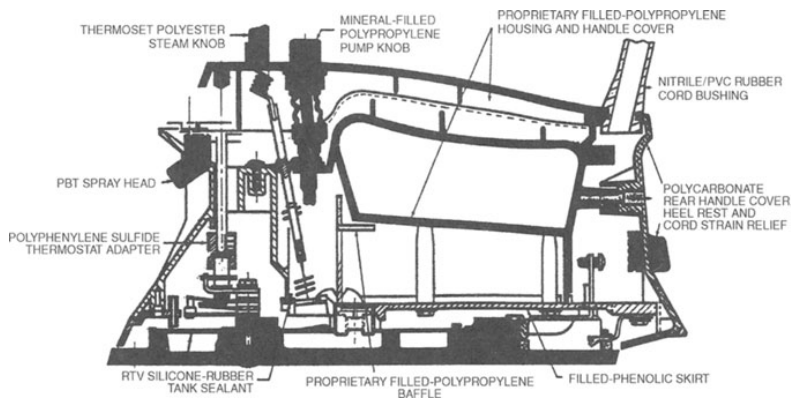


Fig. 4-3 Molded products in an electric pressing iron.

control requirements of the plastic melts. The variables in the machine, plastic melt, and process control must all be managed.

Some information on melt behavior is presented here. See also Chapters 2, 5, 6, and 7 for more processing details.

Basics of Melt Flow

There are variable conditions during molding that influence part performance. Of paramount importance are gate location(s) and controlling the cavity fill rate or pattern. The proper fill helps eliminate part warpage, shrinkage, weld line(s), and other problems or defects (Chap. 8). In the practical world of mold design, there are many instances where tradeoffs must be made in order to achieve a successful overall design. As an example, while a naturally balanced runner system is certainly desirable, it may lead to problems in mold cooling or increased cost due to excessive runner-to-part weight ratios, depending on the production quantity. Software flow analysis guides are available that allow successful designs of runners in which pressure, temperature, rate of flow, etc. are chosen consistently (Chap. 9).

In a typical IMM, the flow of melted plastic into its mold is basically controlled by the injection unit's plasticizing capability and uniformity, control of melt pressure, and screw position. When hot runners are used, their valve gates are involved. Sequential valve gating has become important in some multigate applications. Thus these gates can be opened at different times during injection, increasing control over weldline location and fill balancing. With an open-loop process, the injection unit is tied to the valve gates from the start of the injection by either the time or the screw position. Once the valve is opened, the flow rate is completely controlled by the injection unit. The so-called dynamic-feed closed-loop system uses variable flow valves. Each valve's position is controlled in response to the pressure requirement downstream of the valves.

Mold Filling Hesitation

Here is one of many examples of methods of operating an IMM. To understand the *hesitation effect*, consider the flow patterns throughout injection mold filling. The melt first enters the cavity from the gate, and the flow front reaches the first thin wall section. There is insufficient pressure to fill this thin section, as the melt has an alternative route along the thick section. Melt that just entered the thin section sits there losing heat, until the rest of the mold is filled. When the mold is almost completely filled, the full injection pressure is available to try to fill the thin section. However, the melt in the thin section has frozen, and the thin section is not filled. This problem is caused by the fast–slow–fast (hesitation) filling sequence used. If the melt continues to flow at a nearly steady (uniform) rate, there is no difficulty in filling the thin section. To do so only requires the melt entering the cavity to have the proper temperature, pressure, and rate of injection.

Melt Cushioning

Cushioning the melt means continuing to inject it into the mold cavity under pressure during its shrinkage and until solidification occurs. The purpose is to ensure that, as the stroke is completed and the mold fills, a cushion of melt exists. Usually just a few millimeters (0.04 in.) distance is maintained between the screw or ram tip and the nozzle, which in turn feeds into the cavity or cavities. This action will result in greater compactness and will eliminate or significantly lower the shrinkage of molded products.

Mold Filling Monitoring

Flow-front speed during filling is commonly inferred either from screw position or cavity pressure sensors. The quality of the final molded part, however, is determined by the actual flows of molten plastic into the cavity to pack the melt. The ultrasonic technique is one way of monitoring the filling

DNA Genotek Inc. Exhibit 2029 Page 32
Spectrum Solutions LLC v. DNA Genotek Inc. IPR2023-01424

action. This technology involves the use of ultrasonic transducers and software to verify mold filling patterns and measure flow-front speeds. It permits identifying exactly when mold cavities are filled and switching immediately from injection pressure to packing pressure, saving energy. Ultrasonic beams are emitted from transducers installed on the external surfaces of a steel mold. The beams propagate to the cavity interface. Before the melt arrives at the transducer's position, ultrasonic energy is totally reflected at this interface. After the melt's arrival, part of the beam energy is transmitted into the melt, indicating the arrival. A sensor can monitor the gap caused by the shrinkage of the part away from the mold wall, as well as measure the speed of the gap's development. Ultrasonic waveforms show echoes in the solidifying parts, which can be used to obtain temperature profiles across the melt and to study cooling efficiency.

Sink Marks

Different processing conditions can cause product problems or defects. An example is a sink mark. Sink marks are an indentation on the surface of a molded part that usually occur when there is a significant local change in wall thickness. Examples include ribs, bosses, and undercuts. Sink marks are caused by thermal contraction of the melt during cooling in the mold. Since the volumetric shrinkage of plastics from melt to solid can be about 25% and their compressibility is smaller (perhaps 15%), it is possible to pack out a mold. This action can prevent sink marks during the pressurization phase only. Some compensating flow is necessary to eliminate the sink marks entirely. If it is impossible to use a high enough holding pressure to do so, a lower holding pressure may reduce the marks to an acceptable level.

By analyzing flow as a combination of viscous fluid flow and heat transfer, one can hope to understand what is happening in the mold (Chap. 7). The object is to flow plastic through the thin sections and into the thick sections. With a very slow rate, the pressure drop will be high because of the high heat loss. In the extreme case the plastic can freeze off. With a high holding pressure, there will be a high flow in the pressurization phase and a low flow in the compensating phase. This low compensating-phase flow means that the thin sections will not remain molten long enough for the thick sections such as a boss to be adequately packed out.

Mold Descriptions

Molds are a very important part of the injection molding process, as summarized in Fig. 1-14. There are many different mold designs used to produce all the thousands of different shapes and sizes of products. Examples of a few mold designs are shown in Figs. 4-4 to 4-9.

In the past, when someone purchased an IMM and had made a low estimate of the total cost to set up an operation with its auxiliary equipment, to reduce expenses the mold was skimped. The result most of the time was a disaster, because products did not meet performance requirements or, worse, the cost of molding a quality product went up. The message here is that you get what you pay for.

Molds are of many different designs to meet different product requirements. There are molds that can have common assembly and operating parts so that the tool's cavity or cavities can receive different cavity inserts. Molds can themselves be highly sophisticated and expensive pieces of machinery. They can comprise many parts requiring high-quality metals and precision machining. To take the greatest advantage of these investments, the mold may incorporate many cavities, adding further to its complexity. Many molds have been reengineered as standardized products that can be used with different cavities, runner systems, cooling lines, unscrewing mechanisms, etc.

For over a century it has been easy for those familiar with the engineering (and art) of mold making to obtain the molds they desired. In the past, however, molds were not as complicated as they are now, and in the future

DNA Genotek Inc. Exhibit 2029 Page 33
Spectrum Solutions LLC v. DNA Genotek Inc. IPR2023-01424

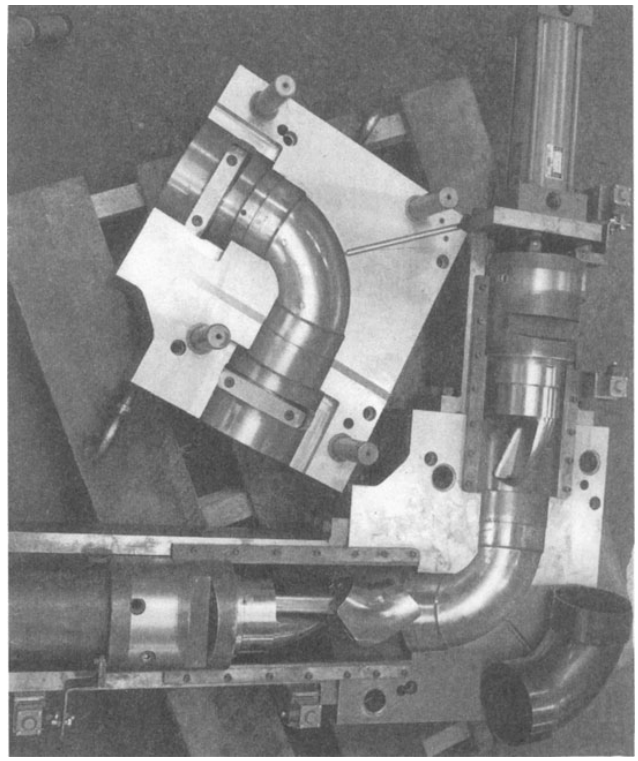


Fig. 4-4 Cavity blocks for pipe elbow are made of prehardened stainless steel; molded elbow is seen in lower right corner.

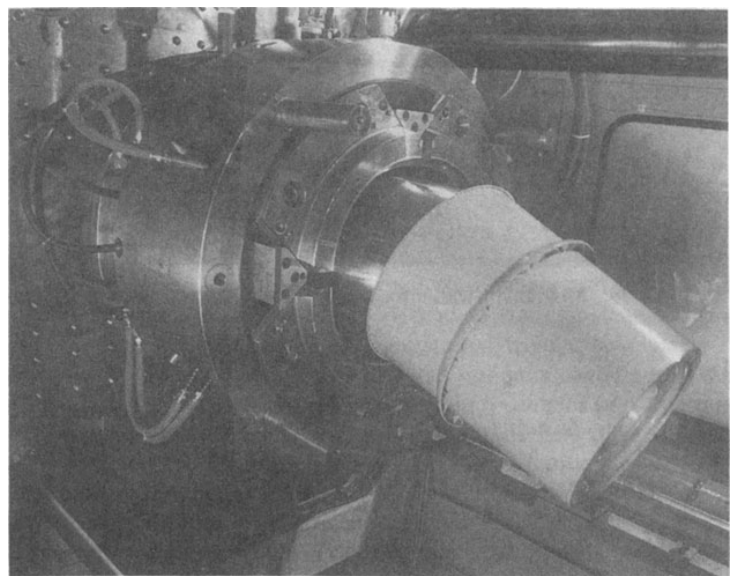


Fig. 4-5 Husky's mold in its 800-ton IMM fabricates plastic drums.

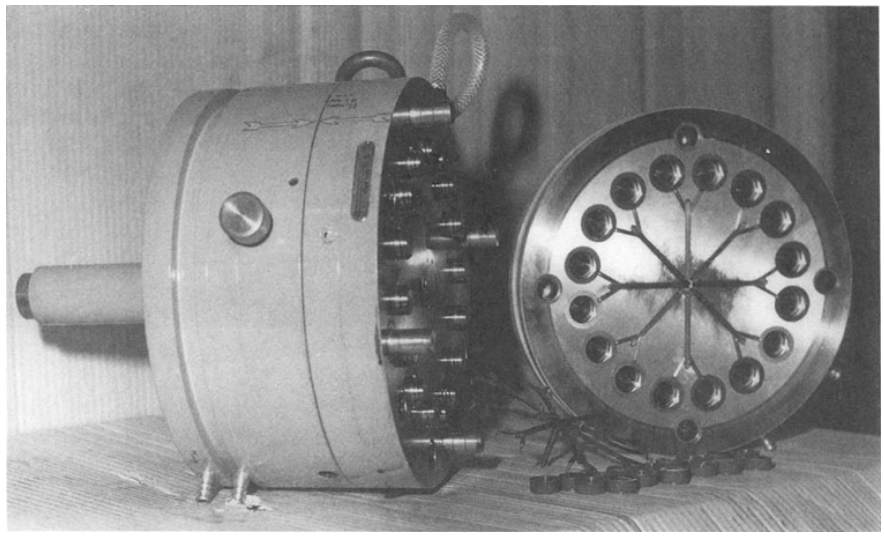


Fig. 4-6 Unscrewing mold for fabricating threaded caps.

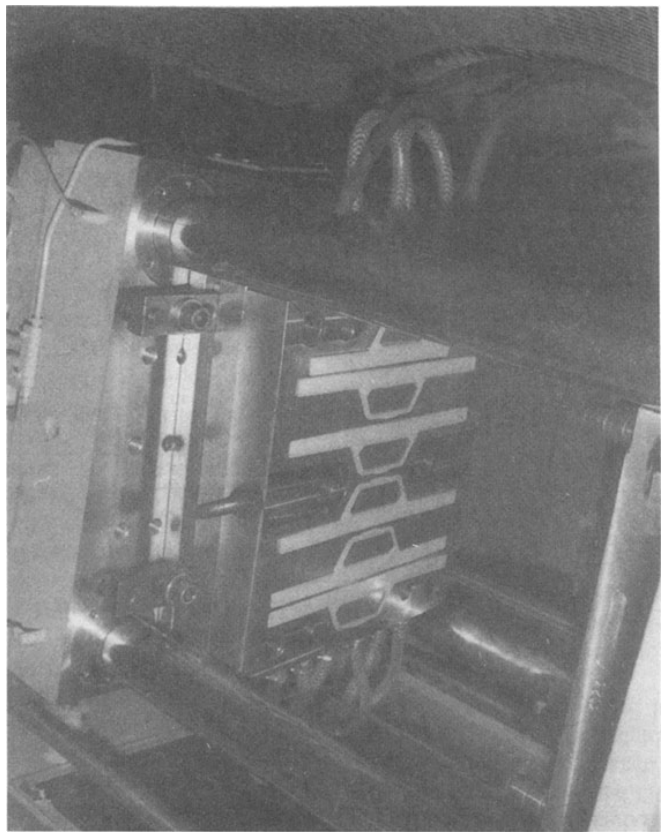


Fig. 4-7 Mold cavity with three sets of handles for plastic bags.

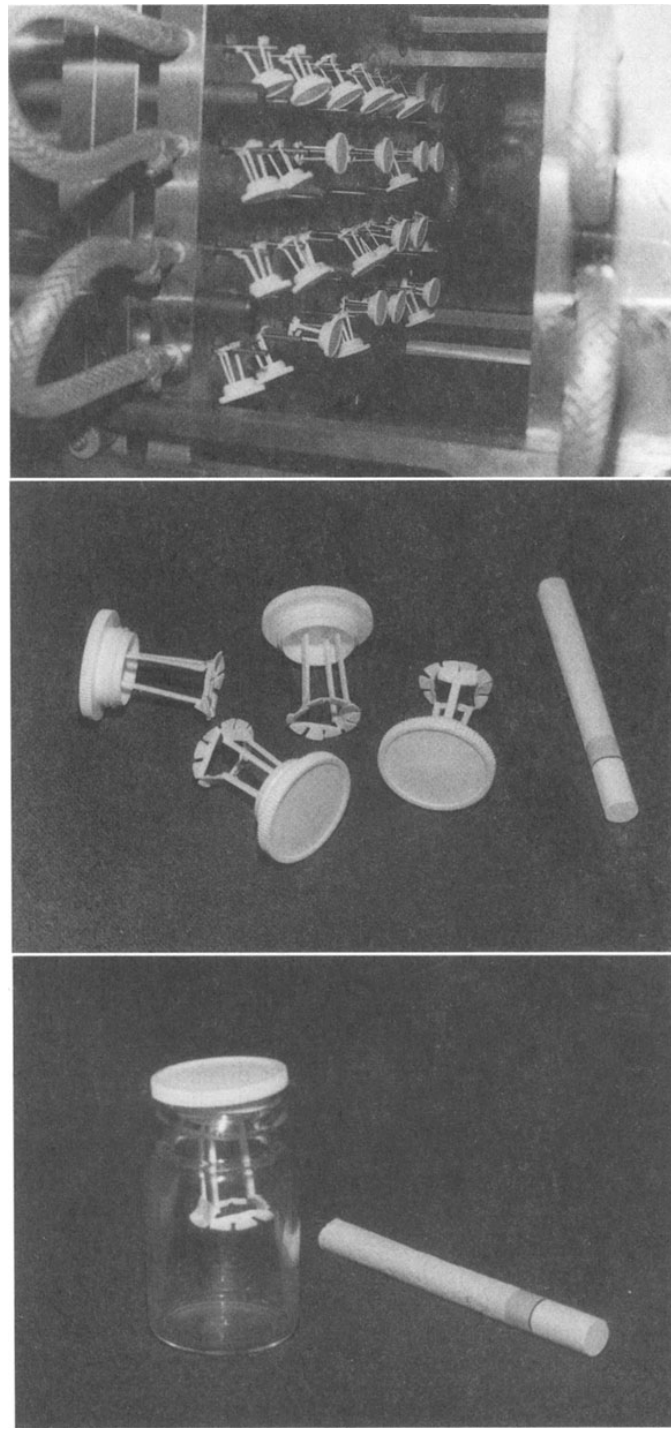


Fig. 4-8 Three views showing flexible insert-type bottle caps made using a simple mold. It permits ejecting parts directly out of the mold without any complex mold movement.

DNA Genotek Inc. Exhibit 2029 Page 36
Spectrum Solutions LLC v. DNA Genotek Inc. IPR2023-01424

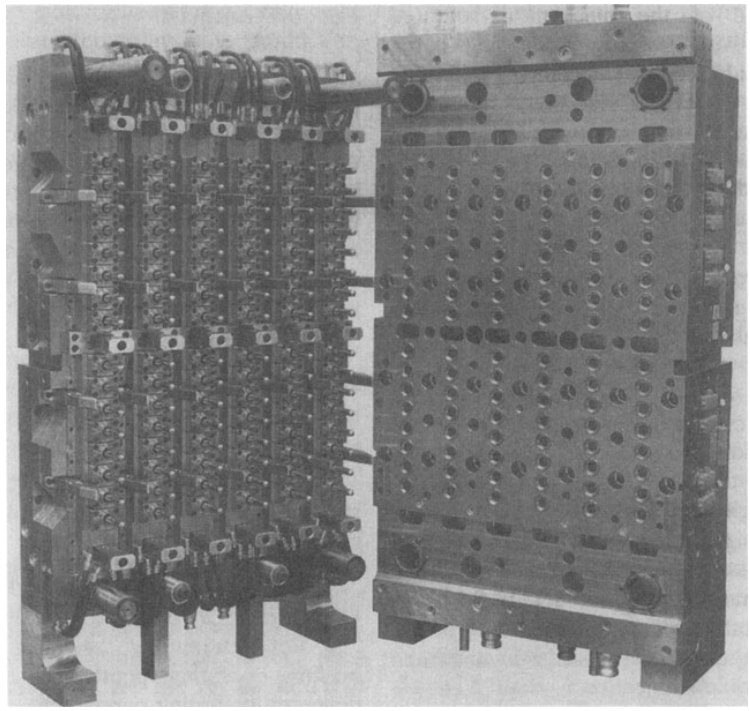


Fig. 4-9 This 96-cavity mold from Husky produces preforms for stretched injection blow molding containers (Chap. 15).

their specification will require even more sophistication. One must either hire qualified persons, who may be difficult to find, or train them in house as described throughout this book. See especially the section on Software and Database Programs.

Mold Basics

The function of a mold is twofold: imparting the desired shape to the plasticized melt and solidifying the injected molded product (cooling for thermoplastics and heating for thermoset plastics). It basically has two sets of components: (1) the cavities and cores and (2) the base in which the cavities and cores are mounted. Figures 4-10 and 4-11 and Table 4-2 show typical layouts and descriptions of products to be molded that include the cavities and cores. Figure 1.11 provides an example of the pressure loading of a plastic melt. Melt moves from injection unit (plasticator), though the mold passageways (sprue, runner, and gate), and into the two cavities.

The mold has two basic parts to contain the cavities and cores. They are the stationary mold half on the side where the plastic is injected, and a moving half on the closing or ejector side of the machine. The separation between the two mold halves is called the parting line. In some cases, the cavity is partly in the stationary and partly in the moving section. The term "mold half" does not mean that the two are dimensionally equal in width.

The size and weight of the molded parts limit the number of cavities in the mold and also determine the machinery capacity required. In the case of large molded parts, such as an auto radiator grille or a one-piece bucket chair, the large exterior dimensions of a single-cavity mold require a correspondingly large clearance between the machine

DNA Genotek Inc. Exhibit 2029 Page 37
Spectrum Solutions LLC v. DNA Genotek Inc. IPR2023-01424

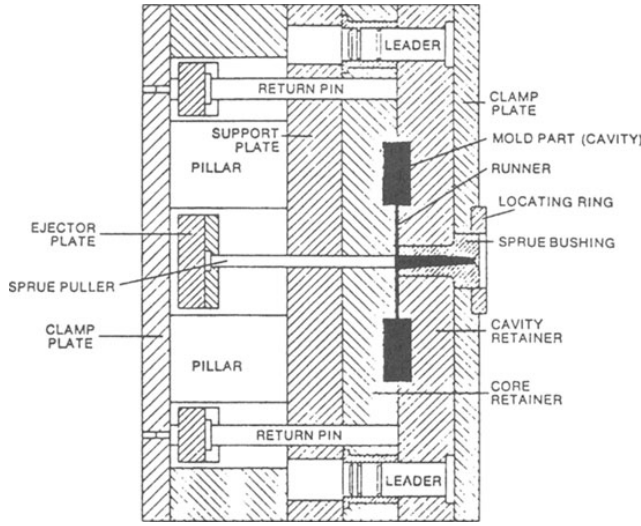


Fig. 4-10 General configuration of a mold.

tie-rods. In turn, the machine tie-rod clearances limit the number of cavities that can be installed in a multicavity mold.

It is important to design a mold that will safely absorb the forces of clamping, injection, and ejection. Furthermore, the flow conditions of the plastic path must be adequately proportioned in order to obtain uniformity of product quality in cycle after cycle. Finally, effective heat absorption from the plastic by the mold has to be incorporated for a controlled rate of solidification prior to removal from the molds.

The mold designer should become thoroughly familiar with the processing information on the plastic material for which the mold is being built. (See Chap. 6 for information on material processing.)

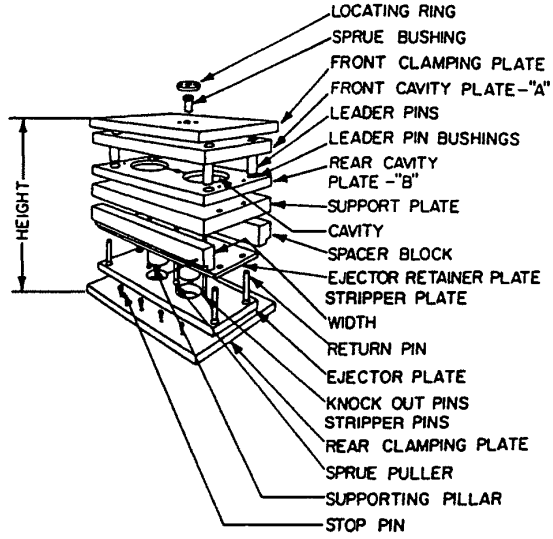


Fig. 4-11 Exploded view of a mold base.

Table 4-2 Functions of the injection mold

Mold Component	Function Performed
Mold base	Hold cavity or cavities in fixed, correct position relative to machine nozzle.
Guide pins	Maintain proper alignment of two halves of mold.
Sprue bushing (sprue)	Provide means of entry into mold interior.
Runners	Convey molten plastic from sprue to cavities.
Gates	Control flow into cavities.
Cavity (female) and force (male)	Control size, shape, and surface texture of molded article.
Water channels	Control temperature of mold surfaces, to chill plastic to rigid state.
Side (actuated by cams, gears, or hydraulic cylinders)	Form side holes, slots, undercuts, threaded sections.
Vents	Allow escape of trapped air and gas.
Ejector mechanism (pins, blades, stripper plate)	Eject rigid molded article from cavity or force.
Ejector return pins	Return ejector pins to retracted position as mold closes for next cycle.

The mold determines the size, shape, dimensions, finish, and often the physical properties of the final product. It is filled through a central feed channel, called the sprue. The sprue, which is located in the sprue bushing, is tapered to facilitate mold release. In single-cavity molds, the sprue usually feeds the polymer directly into the mold cavity, whereas in multicavity molds it feeds the polymer melt to a runner system (cold or hot), which leads into each mold cavity through a gate.

The mold is aligned with the injection cylinder by means of a ring in the stationary mold half, into which the cylinder nozzle seats. The locating ring surrounds the sprue bushing and is used for locating the mold in the press platen concentrically with the machine nozzle. The opening into which the ring fits is made to a tolerance of −0.000 and +0.002 in. (−0.000 and +0.005 cm). The ring itself is made 0.010 in. (0.025 cm) smaller than the opening, providing a clearance of 0.005 in. (0.013 cm) per side. A clearance above this amount may cause misalignment with the nozzle, which in turn would entrap part of the sprue, causing the sprue to stick on the wrong side. The sprue bushing on the locating ring end has a spherical radius of $\frac{1}{2}$ or $\frac{3}{4}$ in. (1.27 or 1.91 cm) to fit the machine nozzle radius. The hole through the length of the sprue has a $\frac{1}{2}$ in./ft taper of $1°11\frac{1}{2}''$ on each side. This hole must have a good reamed and polished finish to prevent sprue sticking.

The parting line is formed by cavity plates A and B. Cavity plate A retains the cavity inserts and supports the leader pins, which maintain the alignment of cavity halves during operation. These guide pins are preferably mounted in the stationary mold half to ensure that the molded product(s) will fall out of the mold during ejection without being fouled. One of the four leader pins is offset by about $\frac{3}{16}$ in. (0.48 cm) to eliminate the chance of improper assembly of the two halves.

The alignment of mold halves is usually accomplished using leader pins. Many moldmakers use tolerances of ±0.0008 to ±0.0013 in. (±0.0020 to ±0.0033 cm) from side pin to bushing. Tighter tolerances of ±0.0004 to ±0.0008 in. (±0.0010 to ±0.0020 cm) provide more accurate alignment and less wear. On ejector systems, a minimum of four leader pins and bushings are used to prevent cocking of the plate, which reduces wear and prevents seizing.

Mating with plate A is plate B, which holds the opposite half of the cavity or the core and contains the leader-pin bushings for guiding the leader pins. The core establishes the inside configuration of a part. Plate B has its own backup or support plate. The B backup plate is frequently supported by

pillars against the U-shaped structure known as the ejector housing. The housing, consisting of the rear clamping plate and spacer blocks, is bolted to the B backup plate, either as separate parts or as a welded unit. This U-shaped structure provides the space for the ejector plate to perform the ejection stroke, also known as the stripper stroke. The ejector plate, ejector retainer, and pins are supported by the return pins. When in an unactivated position, the ejection plate rests on stop pins. When the ejection system has to be heavy because of required large ejection forces, additional supporting means are provided by mounting more leader pins in the rear clamping plate and the bushing in the ejector plate.

The overall height of the mold should correspond to the open space in between the machine platens. In the moving mold half, spacers are used to create space for the ejector system, which consists of two ejector plates with ejector pins. The open space should be such as to permit the ejector pins to complete their ejection stroke. Note that the mold height, or die height, in the usual horizontal operating machine is the horizontal dimension of the mold. When the mold is removed and placed upright on a workbench, its mold height is vertical.

All the mold plates (excluding the ejector parts) and spacer blocks are ground to a thickness tolerance of ± 0.001 in. Conceivably, a combination of tolerances could build up to cause an unevenness at the four corners. If great enough, such a condition would damage a platen when under full ram pressure. It is advisable to check the uniformity of all four corners prior to preparing the base to receive cavities.

Both mold halves are provided with cooling channels filled with coolant to carry away the heat delivered to the mold by the hot thermoplastic polymer melt. For thermosets, electric heaters are located in the mold.

When the mold opens, molding and sprue are carried on the moving mold half; subsequently, the central ejector is activated, causing the ejector plates to move forward, so that the ejector pins push the article out of the mold. Ejector pins have a tendency to produce a very slight flash line, which in some areas of a part may be objectionable; therefore, their location and the amount of recess formed by them in the part should be agreed on with the product designer.

In the smallest injection molding machines, the mold may be completely demountable, and while being filled is held in a simple vise. This can be vertically or horizontally acting to suit the cylinder; some cylinders are downstroking and some horizontally acting. With a horizontally acting cylinder and vertical clamp, the runners and sprue bushing are in the same plane; and often, because the pressures involved are not very great, the hardened sprue bushing is replaced by a simple runner cut into one or both halves of the mold.

With the larger horizontal clamping machines, thought should always be given to whether a horizontal or a vertical flash line is either possible or desirable. In Fig. 4-12 a vertical flash line is shown, whereas Figs. 4-10 and 4-11 depict the more common horizontal flash line. With a mold having a vertical flash line, sometimes called a positive mold, it can be seen that material cannot escape from

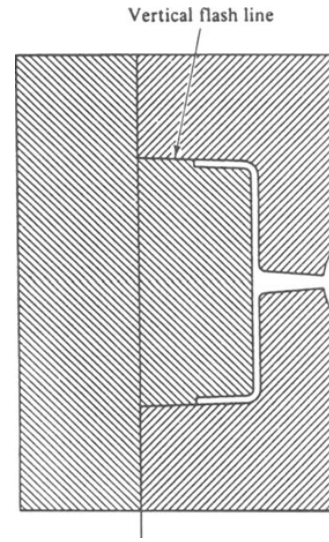


Fig. 4-12 Mold designed with a vertical flash line, typical of a compression mold (also called a positive mold system), where the aligned male and female mold parts meet.

the parting line of the mold until considerable opening movement has taken place. If an overfull shot were to be made, the mold would not flash, although the shot weight might be too great.

This design also has the advantage that oversize moldings can be made with a relatively small mold locking force. If the correct amount of material is injected into the mold, it may open slightly (a few thousandths of an inch), but no material escapes; as the molding cools, the mold closes again, thus *compression-molding* the part. This positive mold is the principal type used in compression molding. It is also the design used with certain structural-foam moldings and in combining injection molding with compression.

Mold Optimization

A mold is a controllable, usually complex mechanical device that must also be an efficient heat exchanger. Hot melt, under pressure, moves rapidly through the mold. Air is released from the mold cavity or cavities to eliminate melt burning, voids in the part, and other defects that would degrade the molded product's performance. In order to solidify the hot melt, water or some other medium circulates in the mold to cool thermoplastics or heat thermoset plastics. Various actions, such as sliders and unscrewing mechanisms, can be used to operate the mold. CAD and CAE programs are available that can aid in mold design and in setting up the complete fabricating process. These programs include melt flow, part solidification, and the meeting of performance requirements.

Optimizing the injection molding process to reach higher productivity requires careful examination of individual components. Compromises in the performance of any one of these can adversely affect productivity. Specifically, overall performance is related to designing the mold for maximum productivity and specifying the machine to obtain maximum output. Machine tools are one of the major investments of the plastics industry. Since its beginnings, the plastics industry has relied heavily on the human skills and experience of moldmakers, and the establishment of strong loyalties, for a good moldmaker means a smooth-running business, and a bad one ruin (1, 7).

A mold is a highly sophisticated piece of machining. It comprises many parts requiring high-quality steels. It also includes cooling channels and possibly hot runner channels for the hot feed of molten plastic. In many cases, it will also contain a number of moving parts, such as ejector pins and moving cores.

To capitalize on the advantages of injection molding, the mold tool may incorporate many cavities, adding further to its complexity. All these parts must function efficiently and smoothly, at high temperature and very high pressure, in a reciprocating machine that may well cycle several times a minute or even more, over long production runs.

Many of these mold elements have been preengineered or standardized and can be incorporated in most tools, whereas the base of the mold may be cast metal. Machined or spark-eroded (EDM) cavities are cut into it. Standard mold sets have been extensively developed, and a solution frequently used for a multicavity mold is first to locate just one cavity on the tool, and then to machine the remaining cavities from the data gained from this first cavity.

Historical information and present activity show that the quality of machine tools used is absolutely critical to the efficiency of molding and the injection molding business. A significant factor in recent years has been the introduction of CNC (computer numerical control) systems for machining all kinds of tools for molding plastics. Although the skill and experience of the toolmaker remain an essential factor, the CNC program gives one precise control over all machining operations. Many of these are lengthy and repetitive, which makes them ideal for computerization.

A further advantage of the computerized control of machine tools is that it allows the whole moldmaking process to be integrated into the product and mold design. Thus, the actual computer tape or disk containing the design data can subsequently be fed into the machine tool control system to

DNA Genotek Inc. Exhibit 2029 Page 41
Spectrum Solutions LLC v. DNA Genotek Inc. IPR2023-01424

give instructions for the detailed work to be done (Chap. 9).

The arrival of CAD (computer-aided design) has had a further benefit in that it brings together all the expertise contributing to the manufacture of molded parts (materials science, product design, moldmaking, and production). In the past, such experts made their contributions individually. This could produce a situation in which most of the work of current experts consists in correcting for the mistakes of previous ones. Within the CAD, computer-aided manufacturing (CAM), computer-aided engineering (CAE), and CNC disciplines, all such experts make their contributions together, virtually at the same time and certainly prior to finalized design.

For maximum productivity, a mold is usually required to operate with the fastest cycle time 24 h/day, 7 days a week. To accomplish this goal, the mold designer must address such areas as cooling, material selection, ejection, access for maintenance, balanced filling of cavities, and the mold's compatibility with the IMM that will be running it. Compromise in any of these areas will reduce the productivity of a mold. This chapter will review in detail the different aspects of moldmaking that are briefly introduced here.

Mold cooling is vital for faster cycles and uniform shrinkage. Cooling is distributed to and from each core and cavity at uniform temperature and pressure to ensure consistent part filling, weight shrinkage, and strength. Optimum cooling is achieved through the turbulent flow of liquid in the channels located as close to the molding surface as possible.

Mold material selection is equally important. As an example, hardened tool steels such as H13, S7, A2, and SS420 can be used to ensure good wear and toughness characteristics. Beryllium copper (BeCu) is used in areas where improved heat transfer would reduce overall cycle time, for example, on gate inserts and core caps. Stainless steel mold plates prevent corrosion and fouling of both water and air lines, thereby improving cooling, reducing maintenance, and extending mold life.

The ejection system can be a major source of wear in a mold. Air ejection of parts is a method often overlooked; a variety of parts ranging from small medicine cups to large industrial containers have been air-ejected. Because there is no mechanical contact, parts can be ejected warmer and cycle time improved. Also, with fewer moving parts, less wear results.

The alignment of individual cores and cavities is necessary because leader pins do not have tight enough tolerances. For example, thin-wall containers requiring concentricity accuracies of ±0.0005 in. (0.0013 cm) and technical parts with stepped parting lines must be protected. When the mold closes, both require individual alignment.

Wear in a mold can be minimized, but it cannot be avoided. Maintenance personnel should be consulted during mold design to ensure easy service and accessibility while the mold is in the IMM. Downtime and expense due to wear can be reduced by making wear items inexpensive and easily replaceable.

In addition to the hot runner, components such as nozzle bands and tips can be replaced when the mold is in the machine by pulling the cavity plate away from the core plate. Downtime for hot runners can also be minimized by using long-life heater bands and manifold heaters. The hot runner system must be reliable and easy to control.

Core and cavity alignment is important. All locations in the mold plates for cores and cavities are usually held to ±0.0002 in. (0.00051 cm). This accuracy ensures the interchangeability of cores and cavities within the plates. Interchangeability can also reduce spare parts inventory.

Computer Systems

This review of mold designs will present basic information on the interactions and integration needed to produce useful molds meeting product requirements. In turn, this information will be useful in understanding the computer operations in mold design described in Chap. 9. Various demands on the mold industry have moved away from

DNA Genotek Inc. Exhibit 2029 Page 42
Spectrum Solutions LLC v. DNA Genotek Inc. IPR2023-01424

production on a strict order basis, whereby manufacturers produce molds according to exact defined parameters submitted by customers, to more of a partnership basis. A moldmaker now becomes involved in a project from the beginning, acting more as a consultant who supports a client throughout all stages of a project.

Taking this into consideration, we see that a moldmaker is not only requested to produce molds, but also has to design the parts according to the requirements of a customer. Furthermore, a moldmaker, in addition to his or her specific technical knowledge, must be acquainted with various plastics and the relevant injection molding technologies that will be used. Usually, the moldmaker has to undertake detailed discussions with the customer and material supplier to decide on the most relevant parameters with regard to tolerances, etc., prior to starting the design process.

An additional need in recent years, to be competitive in the international market where new products have to be presented on shorter time scales, has made the use of computers indispensable. The application of computer systems such as CAD, CAM, or CAE in the mold industry nowadays is a basic requirement. More and more companies are requiring the transfer of design and geometry data by the use of these systems. To work profitably for customers in many areas, it is necessary to install a complete network that directly links computer-generated design data to the numerically controlled production metal-cutting machinery. The CAD system should have two- and three-dimensional capabilities to advance all relevant design activities.

A powerful computer system enables one to react quickly to clients' requirements, especially in terms of changes in design and savings in time and money. Specific software automatically produces the data for CNC programming and forms the connection between design and production departments. The CAM portion of a computer-automated system converts design data into numerical control data used by CNC machines, which will mill and/or erode the shapes onto the parts.

Inevitably, the use of computer systems requires the adjustment of mechanical equipment according to the rapidly progressive CNC technology. It becomes an integral part of the system.

Mold Types

There are many different types of molds, designed to meet many different product requirements (1, 7, 179, 256). Industry generally identifies six basic types for use with thermoplastics. These types are (1) the cold-runner two-plate mold; (2) the cold-runner three-plate mold; (3) the hot-runner mold; (4) the insulated hot-runner mold; (5) the hot-manifold mold; and (6) the stacked mold. Figures 4-13 and 4-14 illustrate these six basic types of injection molds.

A two-plate mold consists of two plates with the cavity and cores mounted in either plate. The plates are fastened to the press platens, and the moving half of the mold usually contains the ejector mechanism and runner system. All basic designs for injection molds have this design concept. A two-plate mold is the most logical type of tool to use for parts that require large gates. This cold-runner system results in the sprue, runners, and gates solidifying with the cavity plastic material.

The three-plate mold is made up of three plates: (1) the stationary or runner plate, which is attached to the stationary platen and usually contains the sprue and half of the runner; (2) the middle or cavity plate, which contains half of the runner and gate and is allowed to float when the mold is open; and (3) the movable or force plate, which contains the molded part and ejector system for the removal of the molded part (Fig. 4-15). When the press starts to open, the middle plate and movable plate move together, thus releasing the sprue runner system and degating the molded part. This type of cold-runner mold design makes it possible to segregate the runner system and the part when the mold opens. The die design makes it possible to use center-pinpoint gating.

DNA Genotek Inc. Exhibit 2029 Page 43
Spectrum Solutions LLC v. DNA Genotek Inc. IPR2023-01424

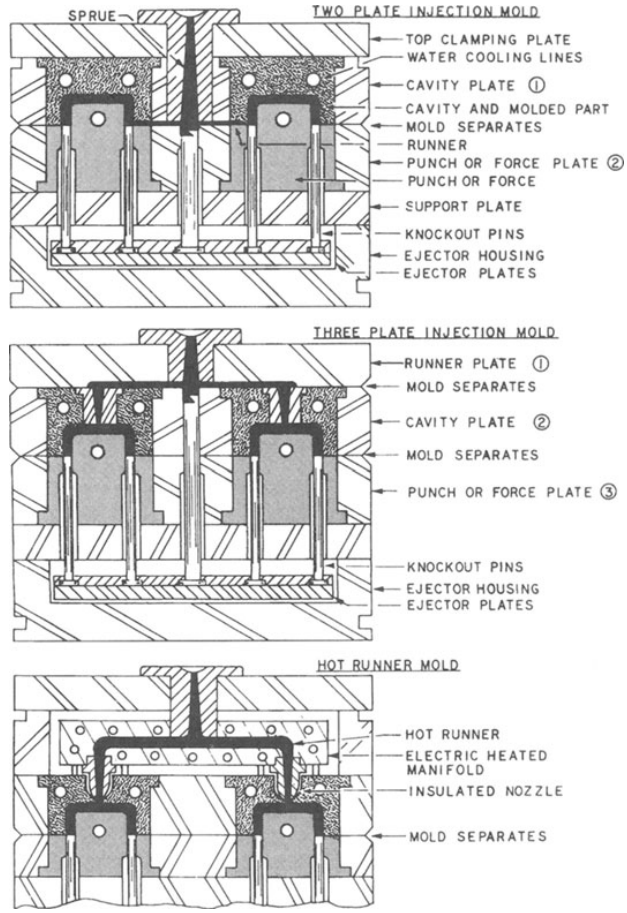


Fig. 4-13 Types of molds, illustrating cold-runner two-plate, cold-runner three-plate, and hot-runner mold for thermoplastics.

In the hot-runner mold, the runners are kept hot in order to keep the molten plastic in a fluid state at all times. In effect, this is a "runnerless" molding process and is sometimes called that. In such molds, the runner is contained in a plate of its own. Hot-runner molds are similar to three-plate injection molds, except that the runner section of the mold is not opened during the molding cycle. The heated runner plate is insulated from the rest of the cooled mold. The remainder of the mold is a standard two-plate die.

Runnerless molding has several advantages over conventional cold-runner-type molding. There are no molded side products (gates, runners, or sprues) to be disposed of or reused, and there is no separating of the gate from the part. The cycle time is only as long as is required for the molded part to be cooled and ejected from the mold. In this system, a uniform melt temperature can be attained from the injection cylinder to the mold cavities. Shot size capacity and clamp tonnage required in the injection molding machine are decreased by the size of the sprue and runners.

The insulated hot-runner mold is a variation of the hot-runner mold (Figs. 4-16 and 4-17). In this type of molding, the outer surface of the material in the runner acts as an insulator for the molten material to pass through. In the insulated mold, the molding

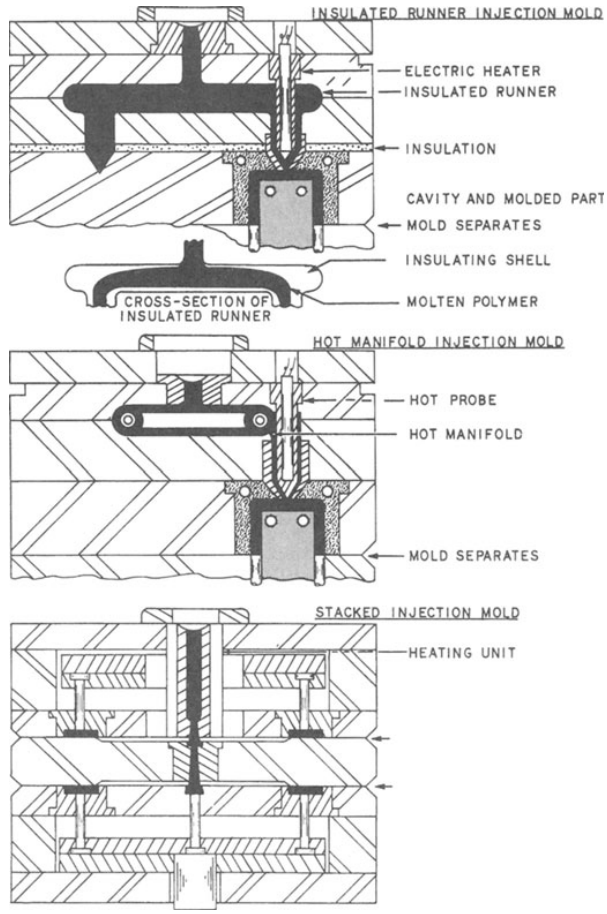


Fig. 4-14 An insulated-runner, a hot-manifold, and a stacked mold.

material remains molten by retaining its own heat. Sometimes, a torpedo and hot probe are added for more flexibility. This type of mold is ideal for multicavity center-gated parts. The diameter of the runner is almost twice that in a cold-runner system.

The hot manifold is a variation of the hot-runner mold. In the hot-manifold die, the runner, and not the runner plate, is heated. This is done by using electric-cartridge-insert probes in sprue, runners, and gates.

Basically, a stacked mold is a multiple two-plate mold, with the molds placed one on top of the other (Figs. 4-18 and 4-19). This construction can also be used with three-plate, hot-runner, and insulated hot-runner molds. A stacked two-mold construction doubles the output from a single press, and requires no more clamping force on the mold if a duplicate set of cavities is used or the maximum clamping cross-sectional area is not exceeded. The machine will require additional shot capacity. Stacked molds are also being used with more than two plates.

Molds For Thermosets

Many mold designs can process thermoset (TS) plastics. A major exception is the insulated mold. By understanding what happens with TSs when compared to thermoplastics

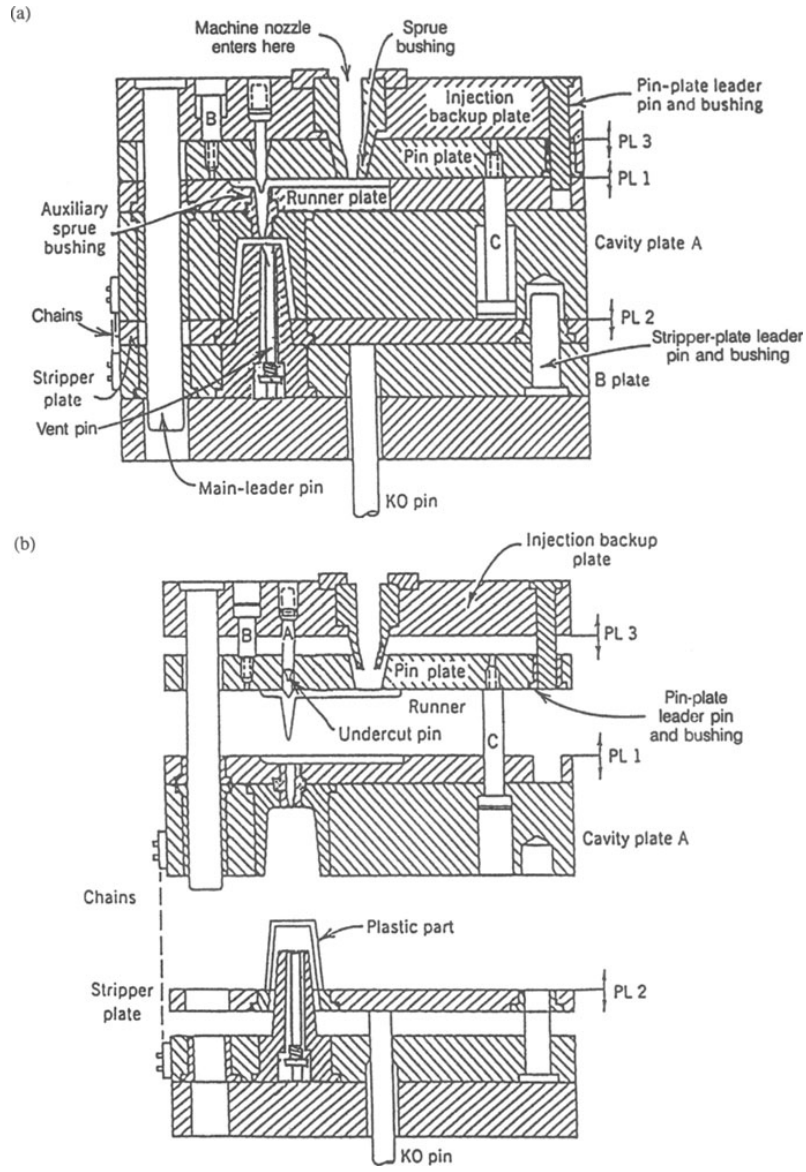


Fig. 4-15 Schematic showing movement of a three-plate mold.

(TPs), one can design a mold specifically for TSs. Throughout this book, as stated in the beginning, most of the discussion here pertains to TPs. When TSs are involved, they will be identified as such, as, for example, later in this chapter when the discussion focuses on hot runners.

In most literature worldwide, when discussing plastics, it is rare that TSs are included, particularly in regard to injection molding and molds. This situation is not unrepresentative, since over 85% (by weight) of all plastics used in all plastics industries are TPs. To date, very few TSs go through

DNA Genotek Inc. Exhibit 2029 Page 46
Spectrum Solutions LLC v. DNA Genotek Inc. IPR2023-01424

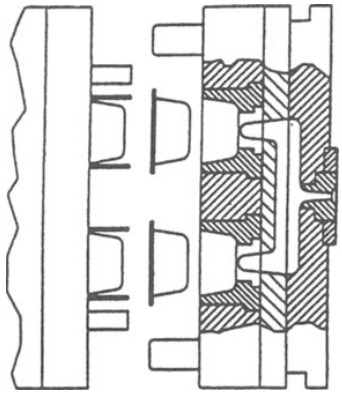


Fig. 4-16 Insulated hot-runner mold with runner removed for easy startup.

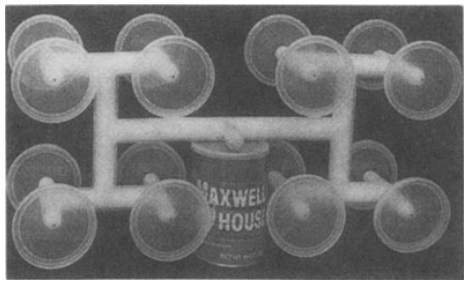


Fig. 4-17 An aggregate of solidified insulated runners with lids attached, removed from its mold (supported on a can).

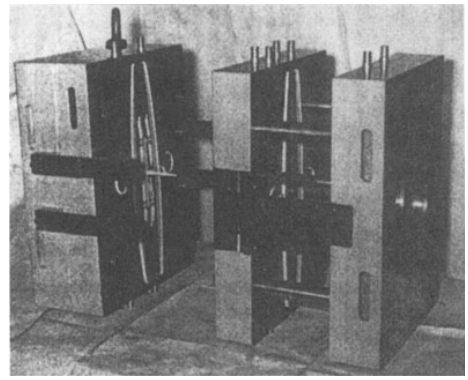


Fig. 4-19 Example of a stacked mold.

injection molding machines; perhaps over 90% are TPs. Most TSs are processed in compression molding, transfer molding, resin transfer molding, reinforced plastics (RP) spray-up, and other RP molding equipment (1, 7).

As explained in Chap. 6 (with more details), the TSs are melted, and after they are injected into the mold, higher temperatures are applied in the mold to solidify or cure (chemically cross-link) the plastics. With sprues and runners subjected to higher heat, they also solidify. This scrap TS material cannot be recycled, since once a TS solidifies, it cannot be resoftened for injection molding.

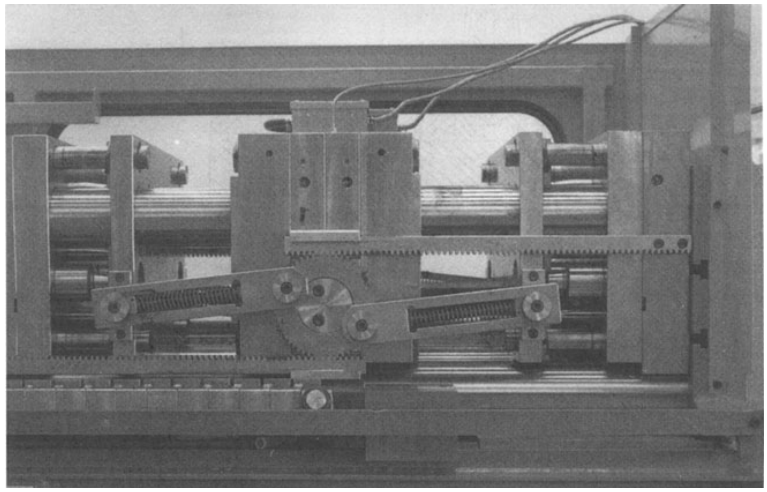


Fig. 4-18 Stacked mold in the open position with rack-and-pinion gears operating the stripper plates.

However, it can be granulated and used as filler in different plastics, particularly TSs. In contrast, TPs after being injected into the mold are simply cooled to solidify them.

Thus, the terminology of melts in the mold is in some respects reversed for a TS. With a TP one refers to a cold runner when the TP solidifies; if it remains liquid, one refers to a hot runner. With TSs, when the runner solidifies, it is a hot runner; but if it remains in a melt phase, it is a cold runner. So the behavior of a TP cold runner is like that of a hot runner for a TS, whereas a TP hot runner is like a cold runner for a TS.

Mold Classifications

The injection molding mold normally is described by a variety of different criteria that include the following:

- *Number of cavities*
- *Material*
  Steel-hardened
  Stainless steel
  Prehardened steel
  Hardened steel
  Beryllium copper
  Aluminum
  Epoxy steel
- *Surface finish*
  Polish
  Chrome plate
  Electroless nickel
  EDM
  Sandblast
  Photoetch
  Impingement
- *Parting line*
  Regular
  Irregular
  Two-plate mold
  Three-plate mold
  Stack mold
- *Method of manufacture*
  Machined
  Hobbed
  Gravity cast
  Pressure cast
  Electroplated
  EDM (spart erosion)
- *Runner system*
  Hot runner
  Insulated runner
- *Gating*
  Edge
  Restricted (pinpoint)
  Submarine
  Sprue
  Ring
  Diaphragm
  Tab
  Flash
  Fan
  Multiple
- *Ejection*
  Knockout pins
  Stripper ring
  Stripper plate
  Unscrewing
  Cam
  Jiggler pins
  Removable insert
  Hydraulic core pull
  Pneumatic core pull

Following these criteria, we may describe a typical mold as follows: a four-cavity, machined, hardened steel, chrome-plated, hot-runner, stripper-plate, tumbler mold.

Plastic Melt Behaviors

The design of molding is governed first by its intended function, and second by the specific limitations of the injection molding process. The properties of the plastic to be used and the engineering aspects of the mold design are added factors.

Consequently, the designing of injection moldings requires not only a thorough knowledge of plastics properties, but also sound insight into the problems of injection molding and mold design. For this reason, close cooperation among the experienced product designer, raw-material supplier, processor, and mold designer is a prerequisite for a product that satisfies the particular requirements of its function and the injection molding process and that can be produced economically. (Other sections of this book underscore the

DNA Genotek Inc. Exhibit 2029 Page 48
Spectrum Solutions LLC v. DNA Genotek Inc. IPR2023-01424

importance of this interdependence of materials, molder, etc.)

Although the injection molding process offers a wide degree of freedom of design, optimum results can be obtained only if the product designer takes the numerous processing factors into account and realizes that design will greatly influence eventual mold construction.

The properties of a plastic part basically depend on how the part is made. Two parts having identical dimensions and made from the same material but molded under different conditions will usually be different parts, with different stress, shrinkage, etc. This means that they will behave differently in service. Thus, the way the plastic flows into the mold and is treated in the mold is of paramount importance in determining the quality of the part.

Cavities are filled from the gate outward. Imaginary lines from the gate to the far sides of the cavities indicate the flow directions. Imaginary lines orthogonal to the flow directions indicate the cross-flow directions. It is important to know these directions because the product properties will vary with the direction, especially in fiber-filled plastics.

Material flows into the cavity because of the pressure gradient applied. As the material reaches the more remote parts of the cavity, the gradient is reduced because of the increased flow length and because the cooling material is becoming more viscous. Eventually, it will freeze and cease to move. If the cavity has not filled by the time this happens, a *short shot*, or incomplete part, results. Clearly, the thickness of the part, pressure gradient, and material viscosity will control the distance of material flow.

Weld and meld lines are created wherever flow fronts meet. They are significant because the properties in the weld- and meld-line regions differ significantly from those in the rest of the part. Because these lines are usually significantly weaker, they become likely points of part failure. Weld lines are created where two flow fronts from opposite directions meet. Meld lines are created where two flow fronts from different but not opposite directions met. Weld lines are weaker than meld lines. When flow fronts meet, the skillful designer will trade off for melding, rather than welding. He or she will also try to ensure that weld and meld lines occur in noncritical regions. These precautions will minimize the risk of part failure.

Whether or not a certain part can be made by injection molding depends first of all on the flow properties of the plastic. Thus, as far as size and shape are concerned, the designer is often faced with certain limitations. Even under optimum molding conditions, very long flow paths, large surfaces, or excessively thin sections may result in short shots.

The extent to which mold cavity dimensions should be larger than the required product dimensions will depend on the total shrinkage of the plastic. For crystalline plastics, total shrinkage may be taken to be the sum of mold shrinkage and after-shrinkage.

Mold shrinkage is the difference between the dimensions of the mold cavity and molding immediately after injection molding and cooling in air. The degree of mold shrinkage depends on the plastic type, processing conditions as they relate to the flow of the melt, and product shape. Moreover, there is a difference between shrinkage in the flow direction of the plastic and shrinkage across that direction. This difference may be substantial, particularly in the case of glass-fiber-reinforced plastics.

The design of a molding must satisfy the functional requirements of the final product, but full allowance must be made for the specific nature of the injection molding process. Curved, grooved, or corrugated surfaces are preferred to flat ones, as the latter are always liable to warp. Warping of flat surfaces can be prevented by means of ribs, but these ribs have a tendency to show up on the other side of the wall as light sink marks. Corners must be rounded, to reduce the risk of notch sensitivity and stress concentrations. Also, rounded corners offer less flow resistance.

In the design of injection moldings, wall thickness should be kept as thin and uniform as possible. This ensures (1) minimum plastic consumption, (2) minimum cycle time resulting from shorter solidification time,

DNA Genotek Inc. Exhibit 2029 Page 49
Spectrum Solutions LLC v. DNA Genotek Inc. IPR2023-01424

(3) uniform shrinkage throughout the molding, (4) uniform mold filling, and (5) minimum risk of internal stresses.

When the design requires differences in wall thickness, the transitions must be gradual. As a general rule, reinforcing ribs must be thinner than the wall they reinforce (about two-thirds of the wall thickness), and their height must not exceed about three times the wall thickness. Wall thickness is governed by not only the functional requirements in service, but also the size of the molding and, more important, the length of the flow path.

The flow of the plastic melt in the mold depends on various factors: plastic used, temperature, mold temperature, length and diameter of sprue and runners, gate type, etc. Together, these factors determine a certain minimum wall thickness. It is understandable that for easy-flow, low-viscosity injection molding materials, the minimum wall thickness that can be filled is smaller than for stiffer-flowing materials having higher viscosity (lower melt index).

Factors differ for practically each different design and plastic, so that an exact specification of minimum wall thickness in relation to flow path is not easily given. However, there is a certain relationship between wall thickness and length of flow path that can be used for most plastics. The length of flow path attainable is proportional to the square of the wall thickness ratio in the range of 0.020 to 0.080 in. of thickness. Thus, if a plastic melt has a flow path of 4 in. with 0.040-in. wall thickness, an increase in the wall thickness to 0.060 in. will increase its flow path to

$$\left(\frac{0.06}{0.04}\right)^2 \times 4 = 9 \text{ in.}$$

Typical flow-path-to-cavity-thickness ratios of general-purpose grades of thermoplastics, based on a cavity thickness of 0.1 in. (2.54 mm) and conventional molding techniques, are given in Table 4-3.

As the material flows through the mold, its condition in the cavity is determined to a major degree by the injection pressure that compresses it into the desired shape. The effective pressure that exerts the densifying force on the molded product is the component that can be recorded in the cavity by a transducer placed, for example, under the head of an injection pin. This cavity pressure component is part of the total injection pressure indicated on the hydraulic machine pressure gauge minus all the pressure drops of the numerous passages (Fig. 1-11).

Table 4-3 Approximate maximum flow-path-to-thickness ratio of thermoplastics

ABS	175 : 1
Acetal	140 : 1
Acrylic	130–150 : 1
Nylon	150 : 1
Polycarbonate	100 : 1
Polyethylene	
Low-density	275–300 : 1
High-density	225–250 : 1
Polypropylene	250–275 : 1
Polystyrene	200–250 : 1
Polyvinyl chloride, rigid	100 : 1

Cold-Slug Well

When we consider the heating condition between the nozzle and sprue bushing, we find a nozzle heated to about the same temperature as the front of the cylinder contacting a relatively cool sprue bushing. As a result, the temperature at the nozzle tip is lower than the required melt temperature. There is a gradual rise in temperature, for about 0.5 to 1 in. into the nozzle, to the normal melt temperature. The TP material lying in the nozzle zone that is not fully up to temperature does not have good flow properties; therefore, if it entered a cavity, it would produce defective parts.

To overcome this situation, a well is provided as an extension of the sprue to receive the cool material, thus preventing it from entering into the runner system (Fig. 4-20). The well is equal in diameter to the sprue at the parting line and is about 1 to 1.5 times the diameter in depth. These sizes may vary considerably, but the important thing is to have the inside of the nozzle of such shape and so heated that the volume of cool material is less than the cold-slug well.

DNA Genotek Inc. Exhibit 2029 Page 50
Spectrum Solutions LLC v. DNA Genotek Inc. IPR2023-01424

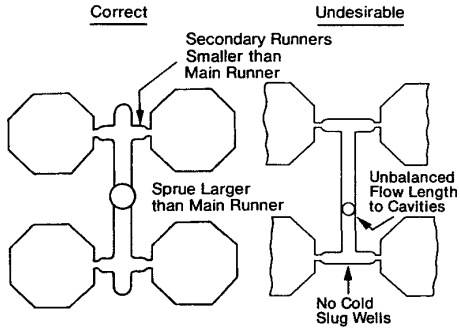


Fig. 4-20 Schematic of a thermoplastic cold runner with cold-slug wells (on ends of runner) and without wells.

In some materials, it is desirable also to have smaller cold-slug wells at the end of the runners or even their branches, to prevent any of the runner-cooled material from getting into the cavity.

A cold slug also performs the function of providing the means of extracting the sprue from its bushing, thereby acting as a retainer for the sprue with runners on the moving half of the mold. During stripping, a pin, which is attached to the stripper plate and also forms the bottom of the well, moves to eject the sprue with runners from the mold. If you are not sure that these wells are unnecessary, plan to leave appropriate space for their inclusion at a later date.

Melt Orientation

Orientation in plastics refers to alignment of its plastic (polymer) chains, whether they are stretched (stressed) or not. High residual stress is not a prerequisite for orientation. Polymer chains have a preferred relaxed state. If they are not frozen so stiff that they cannot move, it is their nature to randomly coil up into a *fuzz-ball* configuration (see Fig. 4-21(a)). When polymer melt is pushed through runners gates and mold cavities, these fuzz-balls distort from the stretching and shearing forces. This distortion creates alignment of chains parallel to each other, as shown in Fig. 4-21(b). This parallel alignment creates strong and weak directions (anisotropy) in a molded part. The situation is somewhat similar to the way the grain in a piece of wood influences how easy it is to break the wood in the grain direction vs. cross-grain [see Fig. 4-21(c)]. Polymers are strong in the orientated direction because the atom-to-atom bonds (such as carbon-to-carbon in ABS) are much stronger than the weak forces attracting neighboring chains. For example, an orientated specimen, broken across the flow direction, can have twice the impact strength of a nonoriented one. Similarly, it is possible for the broken-with-flow strength impact to be only 10% of the broken-across-flow value on a strongly oriented specimen.

Injection molded parts are not uniformly oriented. The degree of orientation varies considerably through the cross section from the surface of the part to the core. It also varies from the gate to the dead end. How pronounced these variations are also depends on the molding conditions—more

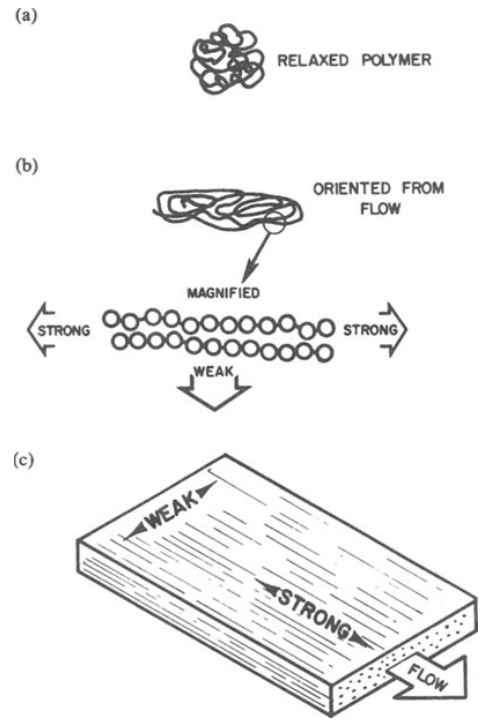


Fig. 4-21 Plastic molecular orientation.

precisely, the point-to-point flow, temperature, and pressure conditions at every location in the mold. It follows from this that certain mechanical properties sensitive to orientation will vary point to point in the molded part. The exact distribution of orientation will determine which properties are affected.

Birefringence All plastics do not exhibit orientation to the same degree. Consider molding a rectangular plaque of clear polystyrene 2 in. (5.1 cm) wide, 6 in. (15.3 cm) long, 0.090 in. (0.229 cm) thick, and gated on the 2-in. end. If the molding were held between crossed Polaroid filters, a colored pattern would be seen. This property is called birefringence and used to measure orientation. The material front that flows past the gate is randomized and freezes in that condition on the walls of the cavity. This section is totally unorientated. However, one end of the molecule is anchored to the wall, and the flow of other material past it pulls the other end of the molecule in its direction, giving a maximum amount of orientation. As the part cools, the orientation is frozen at the walls. The center of the section remains warm for the longest time, allowing Brownian motion to disorient many of its molecules. Therefore, the center section is the least oriented. This is shown by birefringence patterns.

This behavior can be easily demonstrated by milling off one-third (0.030 in. or 0.076 cm) of the thickness. Then one remaining section is highly oriented and the center section, which has been exposed by the milling, is less oriented. If the milled piece is heated, the stretched carbon–carbon linkages should return to their normal position. Because the oriented section has the carbon–carbon linkages lined up more in one direction than they are in the less oriented sections, that part should shrink more. In effect, then, it would be acting like a bimetallic unit, one side shrinking more than the other, and the piece should bend over. This is what happens.

As the amount of orientation depends on the flow and the forces that aid or prevent the motion of the molecular segments, it is easy to see what conditions can affect orientation. Anything that increases the mobility of the segments decreases orientation. Therefore, higher material temperatures, higher mold temperatures, and slower cooling decrease orientation. Pressure on the material limits mobility. Thus, low injection pressures and a short ram forward time decrease orientation. The use of a thicker part would decrease orientation because a longer time would be needed for the center portion to cool with increasing thickness. We shall now examine some practical situations involving orientation.

Practical applications Consider molding a lid or cover 6 in. (15.24 cm) in diameter in a polyolefin. See Fig. 4-22. The shrinkage in the direction of flow is 0.019 in./in. (0.049 cm), whereas the shrinkage perpendicular to flow is 0.012 in./in. (0.030 cm). The difference is caused by the different numbers of carbon–carbon linkages in the direction of and perpendicular to the flow.

Consider a 60° segment of the cover immediately upon molding. Each side will be 3.000 in. (7.620 cm) long. Upon cooling, the two sides in the direction of flow will have shrunk to 2.962 in., and the segments perpendicular to flow will now be 2.976 in. (7.559 cm). A simple trigonometric calculation shows that the central angle is now 60°28′. The full 360° circle is now 362°48′.

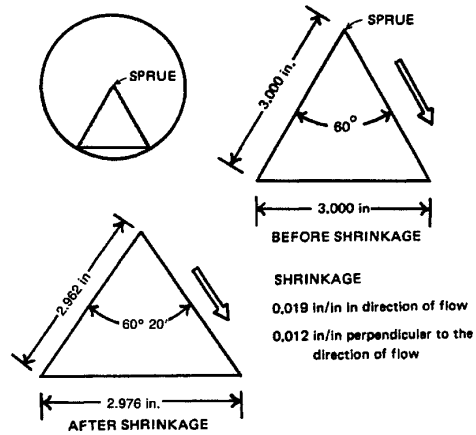


Fig. 4-22 Warping of center-gated polypropylene cover caused by different shrinkage perpendicular and parallel to the direction of flow.

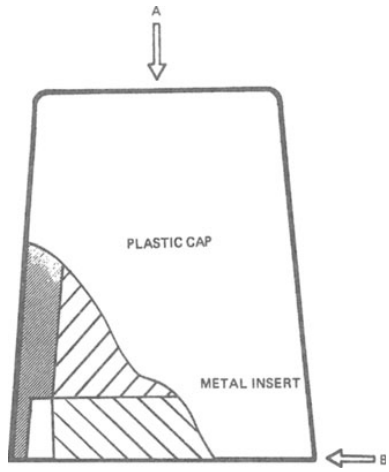


Fig. 4-23 Effect of orientation on a plastic cap molded with a molded insert. Gating at point A will give the cap strength along the walls. Gate B provides strength in the hoop direction.

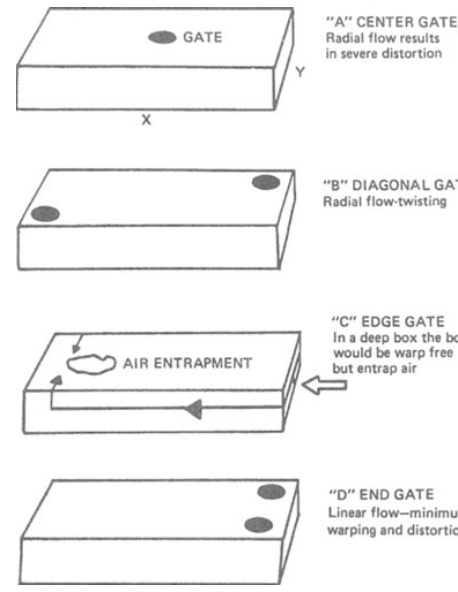


Fig. 4-24 Effect of gate location on a deep molded PE box.

Obviously, the extra material has to go somewhere. If it cannot lie in a plane, it will warp. If the thickness of the material and the ribbing provided enough strength, the part might not visibly warp, but it would be highly stressed. The way to minimize such warp or stress is to mold under those conditions that give the least orientation. Multiple gating also is effective, as is redesigning the cover.

Gate location affects the amount and direction of orientation. Figure 4-23 shows a cap with a metal insert that was used as a protective guard over the fuse mechanism of a shell. The dimensions were controlled by a brass cap, which it replaced. The plastic was molded over a threaded metal insert originally gated at point *A*. After some time in the field, cracking developed around the metal insert. The main strength was in the direction of flow rather than in the hoop (circumferential) direction. Because the thickness of the material could not be increased, the effects of orientation were used by changing the mold and regating at point *B*. The material flowed in the hoop direction and gave the maximum strength there. This slight difference was enough to prevent failure in the field.

Consider gating a deep polyolefin box (Fig. 4-24) using the thinnest possible wall section. Gating the box in the center (A) would give severe radial distortion for the same reasons illustrated in Fig. 4-22. It would be further complicated by the difference in flow length from the gate to point *X* and from the gate to point *Y*. The wall would have to be heavy enough to overcome this stress. Gating it diagonally with two gates (B) would give a radial twist, for the same reasons. It would be much less distorted than the center gate design and allow thinner walls for a stable part. It would, however, require a three-plate mold for the gating.

It would seem logical to gate on the edge of the *Y* portion, as shown in (C). This would be true for a relatively shallow box. With a deep box, however, the material flows around the sides faster than over the top, and air is entrapped somewhere on the top, where it cannot easily be eliminated by venting. This still is not the best method of gating. The preferred method is shown in (D), where there are two gates on the top end of the box. This arrangement gives maximum linear flow without air entrapment and produces a part with the least amount of warp. In most instances, indeed, a satisfactory part could be

DNA Genotek Inc. Exhibit 2029 Page 53
Spectrum Solutions LLC v. DNA Genotek Inc. IPR2023-01424

molded with one gate located on the top end. Another possibility is to place two submarine gates near the top. For large parts, it is sometimes necessary to multiple-gate to ensure even orientation patterns and flow lengths. The main problems encountered in that case are air entrapment and weld lines.

Warpage is the result of unequal stress in the molded part when the stress is strong enough to strain or distort the piece. Warping can be caused by the nature of the material, poor part design, poor mold design, and incorrect molding conditions.

Properties Molecular orientation can be accidental or deliberate. Accidental orientations may be acceptable; however, excessive frozen-in stress can be extremely damaging if parts are subject to environmental stress cracking or crazing in the presence of chemicals, heat, etc. Initially, the molecules are relaxed; molecules in amorphous regions are in random coils, those in crystalline regions relatively straight and folded (Chap. 6). During processing, the molecules tend to be more orientated than relaxed, particularly when sheared, as during injection molding. After heat and pressure are applied and the melt goes through restrictions (molds), the molecules tend to be stretched and aligned in parallel form [Fig. 4-21(b)]. The result is a change in directional properties and dimensions. The amount of change depends on the type of thermoplastic, amount of restriction, and, most important, its rate of cooling. The faster the rate, the more retention there is of the frozen orientation. After processing, parts can be subject to stress relaxation, with changes in performance and dimensions. With certain plastics and processes, the change is insignificant. If changes are significant, one must take action to change the processing conditions, particularly increasing the cooling rate.

By deliberate stretching, the molecular chains of a plastic are drawn in the direction of the stretching, and the inherent strengths of the chains are more nearly realized than they are in their naturally relaxed configurations. Stretching can take place with heat during or after processing. Products can be drawn in one direction (uniaxially) or in two perpendicular directions (biaxially), in which case many properties significantly increase uniaxially or biaxially (Table 4-4 and Fig. 4-25). Film was used for the measurements reported in Table 4-5, since that makes it easier to evaluate orientation properties.

Table 4-4 Effects of orientation on polypropylene films

	Stretch (%)				
Property	None	200	400	600	900
Tensile strength, psi	5,600	8,400	14,000	22,000	23,000
MPa	38.6	58.0	96.6	152.0	159.0
Elongation at break, %	500	250	115	40	40

Property	As Cast	Uniaxial Orientation	Balanced Orientation
Tensile strength, psi (MPa)			
MD	5,700 (39.3)	8,000 (55.2)	26,000 (180)
TD	3,200 (22.1)	40,000 (276)	22,000 (152)
Modulus of elasticity, psi			
MD	96,000 (660)	150,000 (1,030)	340,000 (2,350)
TD	98,000 (680)	400,000 (2,760)	330,000 (2,280)
Elongation at break, %			
MD	425	300	80
TD	300	40	65

MD = Machine direction.
TD = Transverse direction and that of uniaxial orientation.

DNA Genotek Inc. Exhibit 2029 Page 54
Spectrum Solutions LLC v. DNA Genotek Inc. IPR2023-01424

Table 4-5 Effect of molecular orientation on the impact properties of polypropylene films

Material	ASTM Tensile Impact Strength (ft-lb/sq in.)	
	Room Temp.	−20°F (−29°C)
Unoriented PP	40	0
Oriented PP	Above test limit	500

High-Energy Fatigue Impact [55-lb (24.9-kg) weight at 50-in. (127-cm) Height]	
Material	Number of Drops to Failure
Steel	12
Unoriented PP	
41×10^3 psi tensile	1
Oriented PP	
28×10^3 psi, 32% elongation	130

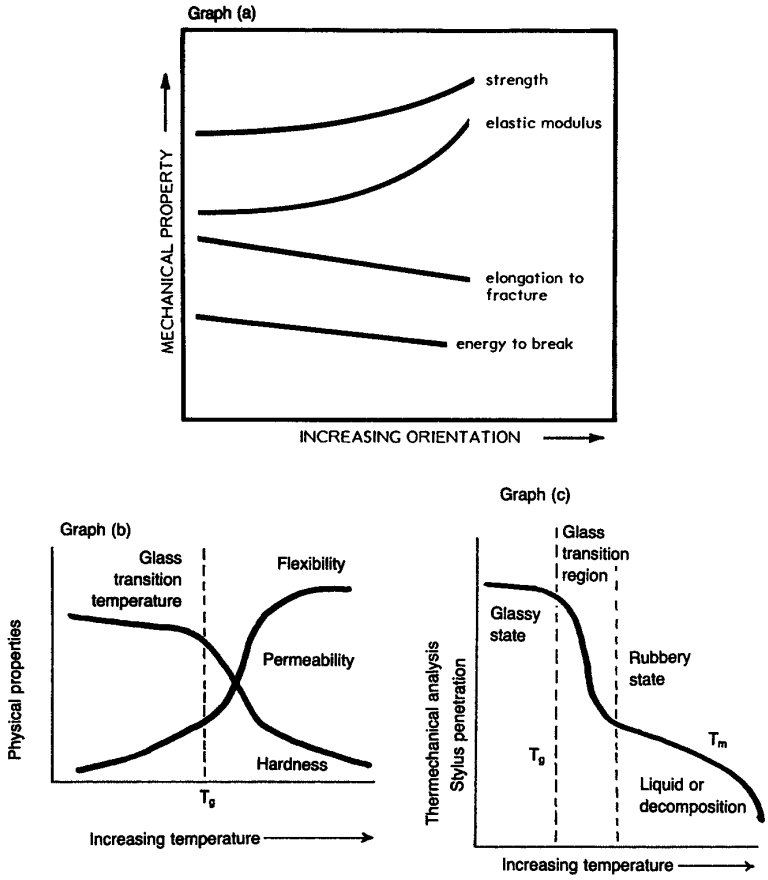


Fig. 4-25 Effect of orientation on the properties of plastics.

Molecular orientation results in increased stiffness, strength, and toughness (Table 4-5), as well as liquid resistance to liquid and gas permeation, crazing, microcracks, and other faults in the direction or plane of orientation. The orientation of fibers in reinforced plastics causes similar improvements. Orientation, in effect, provides a means of tailoring and improving the properties of plastics.

Considering a fiber or thread of nylon-66, which is an unoriented glassy polymer, we observe that its modulus of elasticity is about 2,000 MPa (300,000 psi). Above T_g, its elastic modulus drops even lower, because small stresses will readily straighten the kinked molecular chains. However, once it is extended and has its molecules oriented in the direction of the stress, larger stresses are required to produce added strain. The elastic modulus increases. The next step is to cool the nylon below its T_g without removing the stress, retaining its molecular orientation. The nylon becomes rigid with a much higher elastic modulus in the tension direction (15 to 20×10^3 MPa, or 2 to 3×10^6 psi). This is nearly 20 times the elastic modulus of the unoriented nylon-66 glassy polymer. The stress for any elastic extension must work against the rigid backbone of the nylon molecule and not simply unkink molecules. This procedure has been commonly used in the commercial production of manmade fibers since the 1930s. The major process taking advantage of orientation with injection molding is injection-stretched blow molding (Chap. 15).

Cavity Melt Flow

This section shows how plastic melt flows into the mold cavity. It describes the action occurring where the melt covers the cavity surface and its effect, together with that of core orientation, on the performance of the molded products. It also helps us figure out some useful generalizations on how molding variables affect orientation patterns within the part. The information is generally applicable to plastic behavior; when a specific plastic is described it will be ABS.

Figure 4-26 shows the cross section of a mold cavity where the flow proceeds from left to right. We are looking at a cross section of the part thickness—typically, 0.100 in. thick. The boundary between the advancing melt and still-empty portion of the cavity is called the melt front. This melt front is a stretching membrane of polymer, like a balloon or bubble. Note that the direction of stretching at the front occurs at right angles to the main flow direction. This stretching creates considerable orientation of the polymer molecules. The melt front rolls out like a bulldozer tread onto the surface of the relatively cold mold, creating a zone of surface orientation on the part. There is no evidence, under normal molding behavior, that the melt slides along the cold surface.

Behind the melt front more polymer is flowing—in a sense, to keep the advancing melt front "inflated." In this zone, orientation is caused by the shearing of one polymer

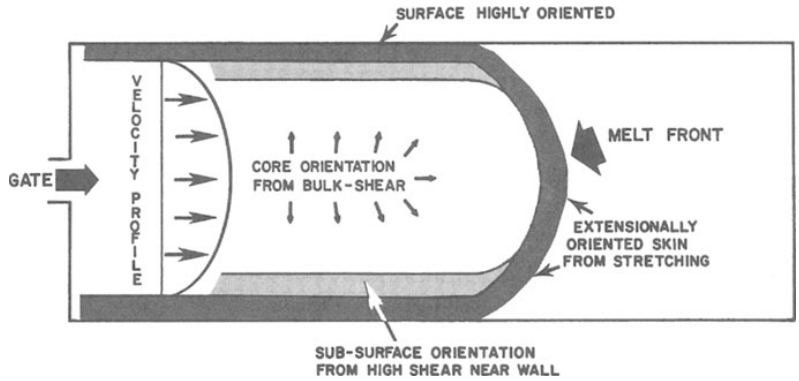


Fig. 4-26 Cavity melt flow model looking at part thickness.

layer over another, which is a consequence of the unavoidable velocity difference resulting from the centerline flowing faster than the edges. This shearing flow creates another band of high orientation just under the surface layer that came from the stretching front. One edge of this band is hung up on the frozen surface layer, whereas the other edge is trying to go along with the main flow. Finally, the core of the part is also oriented to some degree due to shearing and velocity gradations; the orientation gradually diminishes to nothing at the centerline. Thus, cavity flows defines three layers of orientation: surface, subsurface, and core.

Molding variables affect the intensity and relative distribution of the layers because they can influence the two phases of cavity filling. In phase I the melt actually flows into the cavity; phase II involves packing and cooling. Orientation is generated from stretching and shearing during phase I. However, when flow ceases, the stretching and shearing forces essentially disappear, and the polymer orientation can relax out to various degrees. How much relaxation takes place depends on melt temperature, mold temperature, and packing pressure. The net orientation retained in the part is the difference between what was generated during flow minus what relaxed out before the melt cooled down to the freezing temperature.

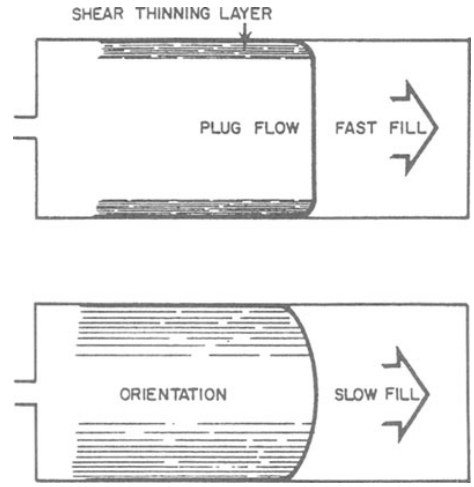


Fig. 4-27 Effect of fill rate.

Fill Rates

Fast fill tends to put more orientation on the part surface and less in the core. This is so because ABS *shear-thins*—mostly near the mold wall where the shear is maximum. As a result, the core *plug-flows*, or slips along under the shear-thinned subsurface layer. This mechanism reduces shear in the core to minimize orientation in the bulk of the part.

Conversely, slow fill minimizes surface orientation and for several reasons allows the core to be more highly oriented than with fast fill (Fig. 4-27). With slow fill there is less shear thinning in the subsurface layer, and the mold has more time to cool the melt while it is flowing into the cavity. These circumstances cause a less locally intense, more evenly distributed orientation through the whole cross section of the part. So fill speed plays a large part in determining *where* in the part cross section the orientation is located: heavily concentrated in a thin layer at the surface, or spread out over the whole core. Fill rate can have additional effects because this variable interacts with melt temperature and packing pressure. Fast fill will cause the melt temperature to rise because of shear heating; slow fill can result in the mold actually cooling the melt. Fast fill also allows better transfer of packing pressure to the melt in the mold, provided that there is a cushion present.

Melt Temperature

Hotter melt yields less orientation in ABS than cold melt for a number of reasons. Hotter melt is less viscous, so the stretching and shearing forces that generate orientation are reduced. Hotter melt also freezes more slowly and allows more time for melt relaxation (orientation decay) after flow ceases and before the part sets up. Figure 4-28 shows the combined response of fill rate and melt temperature on surface and core orientation.

Mold Temperature

Generally, mold temperature has a weaker influence on orientation than fill rate or melt

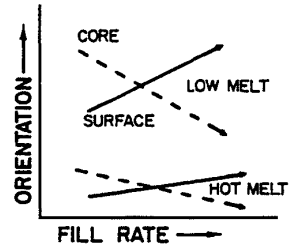


Fig. 4-28 Effect of melt temperature and fill rate on orientation.

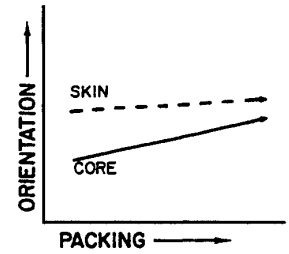


Fig. 4-30 Effect of packing pressure on orientation.

temperature. There is little evidence that mold temperature has much effect on surface orientation. A hotter mold does tend to reduce core orientation because the melt freezes more slowly, allowing more time for orientation relaxation (7) (Fig. 4-29).

Packing Pressure

If a cushion is present and injection hold time is sufficient, increased packing pressure generally increases orientation for two reasons. Creeping flow can occur during packing to compensate for the cooling and shrinking melt in the cavity (5). This slow creeping flow creates core orientation, particularly near the gate. Higher pressures also can reduce melt relaxation, so that more of the fill-induced orientation is retained (Fig. 4-30).

Mold Geometry

The geometry of the mold cavity can also influence the cause and effect links.

- For a given injection rate (ram travel or volumetric flow through the sprue), the local melt front velocity (MFV) will be higher for a thin part than a thick one. Thus, local downstream variations in a part's wall thickness cause MFVs to change just as if the ram speed were being varied throughout the shot.
- A part with two gates can have one-half the local MFV of a part with one gate.
- Undersize runners and gates create more shear heating, and thus higher melt temperatures.
- Thick parts cool more slowly than thin ones, providing more opportunity for relaxation of core orientation.
- The greater the distance from the gate, the lower the local packing pressure.
- Small perturbation in cavity surface geometry can also have curious but important effects on part surface properties, such as electroplate adhesion and paint soak.

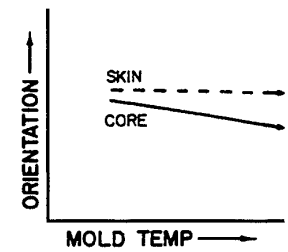


Fig. 4-29 Effect of mold temperature on orientation.

Flash Guide

While the cause of flash may seem elementary, its cure is not. Understanding temperature, cavity pressure, and timing is a good start on a long-term fix. Basically flash is caused when the pressure of the plastic melt is greater than the clamp holding pressure. The basic problem can be with the plastic, IMM, controls, and/or mold. The viscosity of the plastic can have a lot to do with flash. Less viscous melt will seep into the slightest crack at the parting line and act as wedge to force the mold halves apart. An important aspect is temperature, since it directly effects viscosity. The higher it goes, the more fluid the melt; the lower, the more viscous.

The external pressure must be sufficient to fill the cavity, compress the melt, and

DNA Genotek Inc. Exhibit 2029 Page 58
Spectrum Solutions LLC v. DNA Genotek Inc. IPR2023-01424

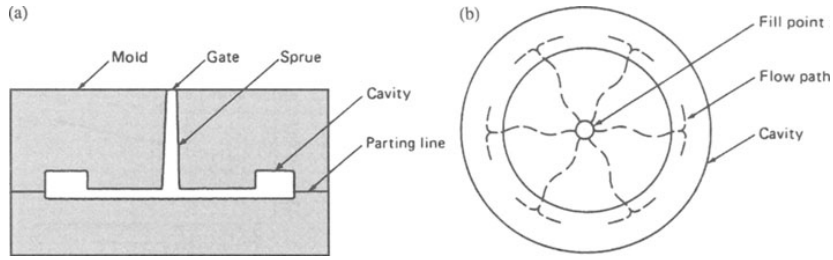


Fig. 4-31 Plastic melt does not flow uniformly through the diaphragm of the plate mold (a) in the compensation phase, but spreads in a branching pattern (b).

compensate (by packing) for the melt's shrinkage which may be just a few percent or for certain plastics as much as 25%, as it goes from liquid melt to solid (Chap. 4).

Molding Variables vs. Performance

As described throughout this book, there are variables during molding that influence product performance (Chap. 8; Chap. 11, subsection on Plastic Material and Equipment Variables; etc.). The information presented here shows how melt flow variables influence product properties in the context of this chapter's concern with manufacturing techniques. A flow analysis can be made to aid designers and moldmakers in obtaining a good mold (Chaps. 5, 7, 8, and 9). Of paramount importance is controlling the fill pattern of the molding so that parts can be produced reliably and economically. A good fill pattern for a molding is usually one that is unidirectional, thus giving rise to a unidirectional and consistent molecular orientation in the molded product. This approach helps avoid warpage problems caused by differential orientation, an effect best exemplified by the warpage that occurs in thin center-gated disks. In this case, all the radials are oriented parallel to the flow direction, with the circumferences transverse to the flow direction. The difference in the amounts of shrinkage manifests itself in warpage of the disk.

In order to achieve a controlled fill pattern, the mold designer must select the number and location of gates that will result in the desired pattern. Flow analysis can help by allowing the designer to try multiple options for gate locations and evaluate the impact on the molding process. This analysis often can be conducted with the product designer to achieve the best balance of gate locations for cosmetic and molding considerations. Figures 4-31 to 4-38 show various flow patterns, orientation patterns, and effects on property performance. (See also Chaps. 4 and 8.)

In the practical world of mold design, there are many instances where design tradeoffs must be made in order to achieve a successful overall design. Although naturally balanced runner systems are certainly desirable, they may lead to problems in mold cooling or increased cost due to excessive runner-to-part weight ratios. Additionally, there are

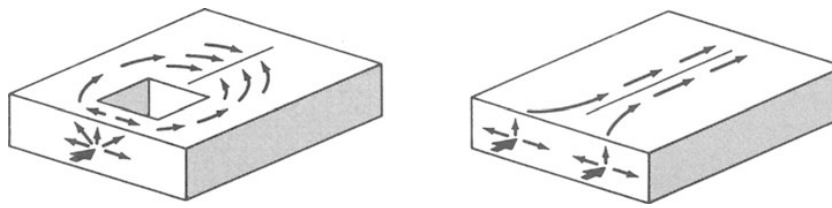


Fig. 4-32 Flow paths are determined by part shape and gate location. Flow fronts that meet head on will weld together, forming a so-called weld line. Parallel fronts tend to blend, usually resulting in a less distinct weld line with a stronger bond.

DNA Genotek Inc. Exhibit 2029 Page 59
Spectrum Solutions LLC v. DNA Genotek Inc. IPR2023-01424

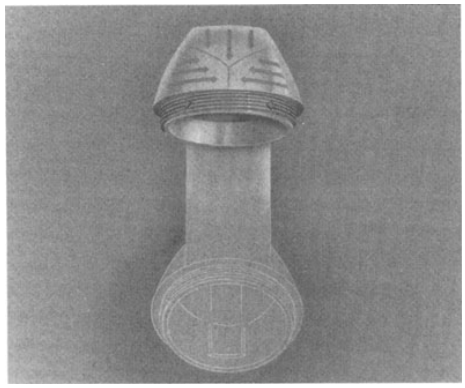


Fig. 4-33 Example of flow with weld lines in a type of telephone handset where the gate was located at the top center of the handle.

many cases, such as parts requiring multiple gates or family molds, in which balanced runners cannot be used. Flow analysis tools allow successful designs of runners to balance for pressure, temperature, or a combination of both.

Shot-To-Shot Variation

During injection molding, shot-to-shot variations can occur. Major causes of inconsistency are worn nonreturn valves, bad seating of a nonreturn valve, a broken valve ring, a worn barrel in the valve area, or a poor heat profile. To identify the cause, one follows a logical procedure. Any problem caused by the valve will cause the screw to rotate in the reverse direction during injection. To locate the trouble, one must pull and inspect the valve, and check the outer diameter (OD) of the ring for wear. The inspector looks for a broken valve stud (caused by cold startup when the screw is full of plastic), bad seating of the ring or ball [angles of the ring inner diameter (ID) and seat must be different, in order to ensure proper shutoff action at the ID of the ring], and a broken ring. One checks the dimensions of the valve and compares them with those determined before using the machine.

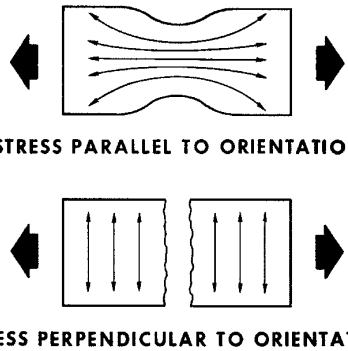


Fig. 4-34 Effect of orientation on strength. The highest tensile strength is in the direction parallel to the orientation.

A poor heat profile for crystalline resins can cause unmelted material to be caught between the ring and seat, holding the valve open and allowing leakage. A change in the heat profile or the machine's plasticizing capacity is not sufficient to correct the problem. For any resin, if the problem does not occur with every shot, the cause may be improper adjustment or damaged barrel heat controls.

Nonuniform melt density could be caused by nonuniform feeding to the screw and/or the regrind blend, which could have a different bulk density. Increasing the back pressure may help. This throughput condition, the residence time of the plastic in the barrel, and the barrel heat profile are all important in obtaining the best melt quality. The heat quality is the most important parameter and varies from resin to resin, as well as with different cycle times and shot sizes. As the following example shows, a screw operating under two different conditions will produce different results.

Consider a screw with a 2-in. (5.1-cm) diameter, 20/1 *L/D*, and 20-oz (0.57-kg) melt screw capacity. With a 15-sec cycle and shot size of 2 oz, it operates as follows:

20 oz (screw capacity) ÷ 2 oz = 10 cycles
15-sec cycle ⇒ 4 cycles/min
10 cycles ÷ 4
= 2.5 min of residence time, from the time plastic starts through the screw until it enters the mold.

Another set of requirements uses a 6-oz (0.17-kg) shot size with the same 15-sec cycle:

20 oz ÷ 6 = 3.33 cycles
3.33 cycles ÷ 4 = 0.83 min of residence time

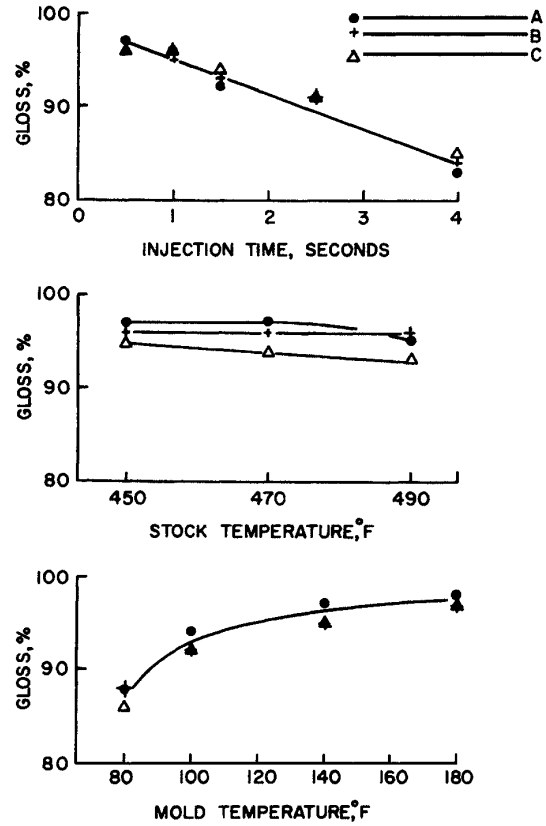


Fig. 4-35 Effect of molding conditions on the gloss of an ABC plastic.

In the second case, a higher rate of melting will be required, with the probability that the screw will be inadequate for the melt, and problems will develop.

The inventory in a screw will run between $1\frac{1}{2}$ and 2 times the maximum shot size rating in polystyrene. With other resins, calculate the differences in density to arrive at the maximum shot size and expected inventory.

Cavities

Cavity Melt Flow Analyses

The purpose of flow analysis is to gain a comprehensive understanding of the mold filling process. There are more and less sophisticated software models that provide detailed information concerning the influence of mold filling conditions on the distribution flow patterns and flow vectors, shear stresses, frozen skins, temperatures, pressures, and other variables; the less sophisticated provide fewer variables, but they may be the only ones required. From these data, conclusions regarding expected tolerances as well as part quality strength, appearance, weld lines, etc. can be drawn. The likelihood of warping surface, blemishes, and strength reduction due to high shear stress can be anticipated. From this analysis, the best and practical mold filling conditions can be selected.

Cavity Melt Fountain Flow

The melt pattern entering the cavity forms a fountain (stretching balloon) (Fig. 6-41). The stretching, oriented outer surface of the

DNA Genotek Inc. Exhibit 2029 Page 61
Spectrum Solutions LLC v. DNA Genotek Inc. IPR2023-01424

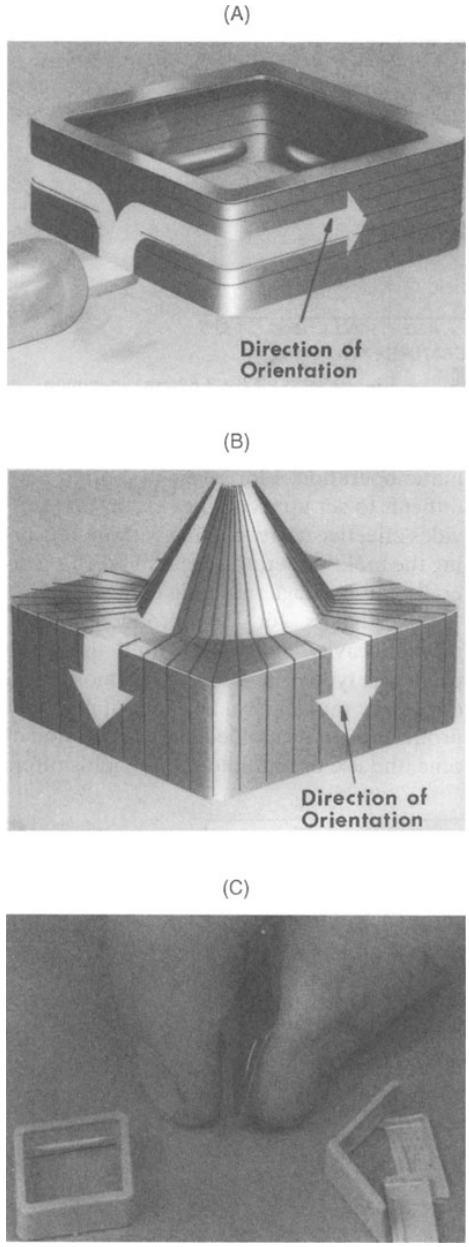


Fig. 4-36 Example of locating a gate to obtain the required performance of a retainer-molded product that is subject to being flexed in service: (a) Edge-gated retainer. (b) Center-gated retainer. (c) Left retainer and middle retainer (between fingers), which were edge-gated, did not fail on repeated flexing, but the center-gated retainer on the right side failed on initial flexing.

melt front covers the inside wall of the cavity. The melt that follows mainly fills within the fountain flow. The result is a nonuniform orientation in the cross section of the molded part; however, the part may still meet performance requirements. The degree of ballooning or bubble formation is controllable so that specific desired properties can be obtained.

Cavity Evaluation

Once the plastic has been selected and the design of the product finalized, a decision must be made as to whether a single- or a multiple-cavity mold should be used. Points to be taken into consideration include:

1. Number of moldings and period of delivery
2. Quality control requirements (dimensional tolerances, etc.)
3. Cost of the moldings
4. Polymer used (influencing location and type of gate)
5. Shape and dimensions of molding (influencing position of mold parting line and mold release)
6. Injection molding machine (determining shot capacity, plasticizing capacity, and mold release)

It is logical that the decision will aim to ensure economy of production; however, there should be a sufficient guarantee of the quality of the product. Advantages and disadvantages must be weighed carefully.

Advantages of single-cavity molds are:

1. Their simple and compact construction, leading to lower cost and quicker construction then for multicavity molds.
2. That the shape and dimensions of moldings are always identical. In multiple-cavity molds, it can be extremely difficult to make intricate cavities exactly alike. Consequently, if technical articles are to be produced within very close dimensional tolerances, a single-cavity mold often may be preferred.

DNA Genotek Inc. Exhibit 2029 Page 62
Spectrum Solutions LLC v. DNA Genotek Inc. IPR2023-01424

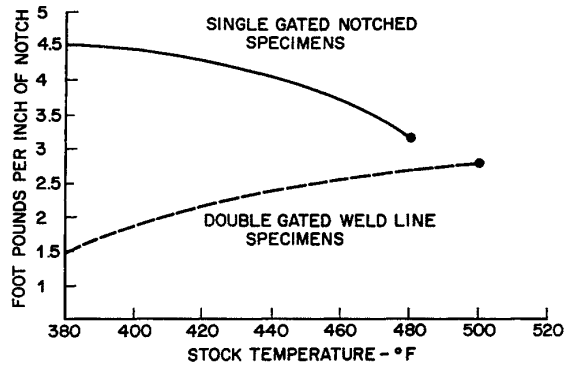


Fig. 4-37 Izod impact strength of ABS plastic [$\frac{1}{3} \times \frac{1}{2} \times 3$-in. ($0.85 \times 1.27 \times 7.62$-cm) specimen].

3. Their better process control, since processing conditions need only be adjusted to suit one molding.

4. That single-cavity molds allow greater latitude in design, for both product and material. The technical requirements regarding gating system, ejector system, cooling system, and mold parting line can, in the majority of cases, be met without compromise.

The complexity of multiple-cavity molds not only makes such molds expensive, but also increases the risk of faults in fully automatic operation. Moreover, it is often very difficult to set up a cooling system that provides effective cavity cooling without impairing the mold's operating reliability. This generally causes longer molding cycles.

Nevertheless, for long production runs, multiple-cavity molds are often the more profitable type (Fig. 4-39). (Most molds produced are multicavity). If large numbers of products must be molded in a short period of time, the use of multiple-cavity molds offers

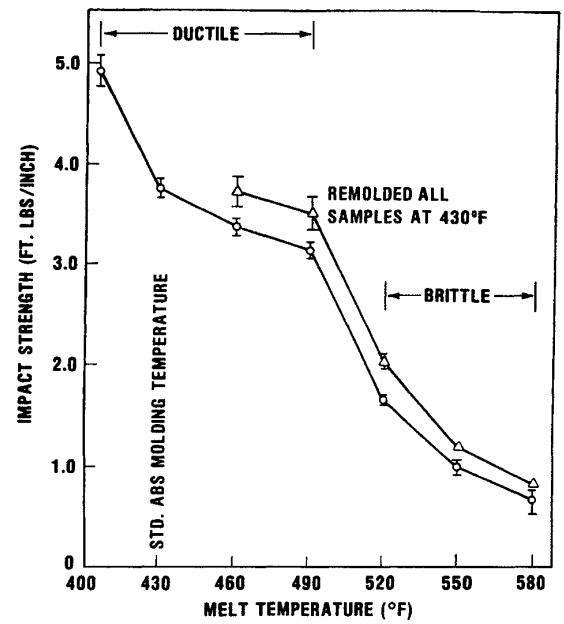


Fig. 4-38 Impact strength vs. melt temperature in white ABS plastic.

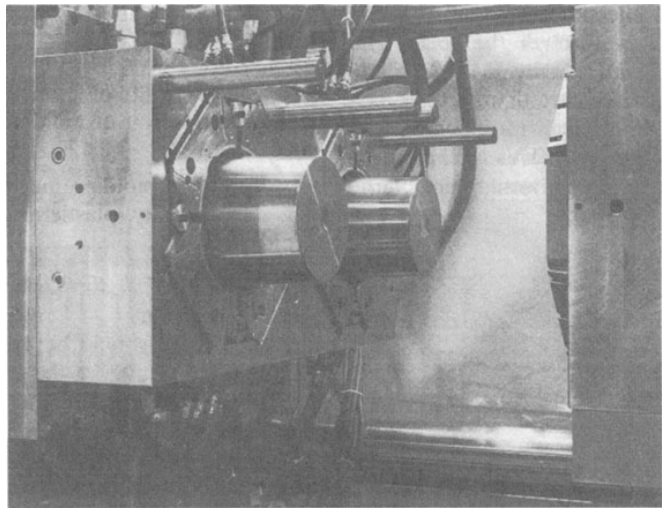


Fig. 4-39 Husky IM system's two-cavity mold with square lock design for a 5 USG pail (910 g).

distinct advantages. If very small articles must be molded and no suitable machine is available, a multiple-cavity mold is the only possibility.

As a starting point, the number of cavities must be established. This is usually determined by the customer, who balances the investment in the tooling against part cost. From the molder's point of view, the number of cavities can be determined as follows:

1. The maximum number of mold cavities follows from the ratio of the shot weight S and to the molding weight W including sprue and runners. S is generally taken to be 80% of the shot capacity of the machine. If S is 200 g and W 50 g, the maximum number of mold cavities is

$$\frac{S}{W} = \frac{200}{50} = 4$$

2. Additionally, the number of cavities is governed by the plasticizing capacity P of the machine and estimated number of shots per minute, X. If P is 18 kg/h = 300 g/min and X is 2, the number of cavities is

$$\frac{P}{X \times W} = \frac{300}{2 \times 50} = 3$$

In this example, the mold should not contain more than three cavities.

Other factors also affect the number of mold cavities, and the location of the mold cavities is also subject to restrictions. One limitation is that the distance between the outer cavities and primary sprue must not be so long that the plastic melt loses so much heat in the runners that it is no longer sufficiently fluid to fill the outer cavities properly. Speed of filling tends to minimize this, provided that the melt viscosity and runner cross section are adequate, and the gates are not too restrictive.

The layout of the mold, dictated by product design, may also restrict the number of cavities, so that the capacity of the IMM cannot be fully used. For instance, if side actions are used, the cavities must, of necessity, be situated in one or more parallel rows.

One of the most important aspects of multiple-cavity mold design is the layout of the feed to the cavities. The cavities should be so arranged around the primary sprue that each receives its full and equal share of the total pressure available, through its own runner system (so-called balanced runner system). This requires the shortest possible distance between cavities and primary sprue, equal runner and gate dimensions, and uniform cooling. When practical, a correct arrangement of cavities will avoid differences

DNA Genotek Inc. Exhibit 2029 Page 64
Spectrum Solutions LLC v. DNA Genotek Inc. IPR2023-01424

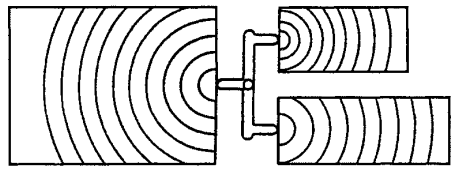


Fig. 4-40 Multicavity family mold with different size cavities.

in product dimensions, stress buildup, mold release problems, flash, etc.

Multiple-cavity molds preferably should contain cavities of identical shape. In principle, different parts of an article should not be produced by means of one multiple-cavity mold, although this is sometimes done for reasons of economy. In that case, the largest cavities, as in a family mold (Fig. 4-40), should be nearest to the sprue, and the runner and gate dimensions should be checked by test molding. If necessary, corrections should be made by balancing the feed to each cavity first by using appropriate runner sizes; this subject will be discussed later. Figure 4-41 shows balanced and unbalanced cavity layouts. The lower left layout in Fig. 4-41 has three runners, which divide and balance the cavities. This arrangement provides a flow pattern with cold-slug wells at the ends of the runners that serve to trap partly cooled plastic before it enters the cavities.

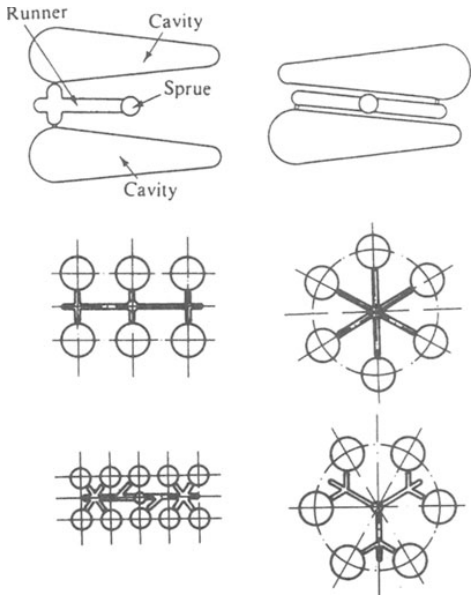


Fig. 4-41 Examples of cavity layout in multicavity molds. Right views show balanced systems. Left views show unbalanced systems except for top left, which is balanced.

It is not good practice (although it is very often done) to have cavities of greatly differing size in a single mold. Many so-called family molds require this, and almost without exception they cause difficulties. It may be that some of the moldings warp, whereas others show excessive frozen-in strain, and any attempt at balancing for melt flow and cooling the cavities leads to long and tedious trials.

Machine Size

A consideration for the optimum number of cavities is the size of the IMM in which the mold will be run. Of course, the ideal approach is to design the mold to meet part and cost requirements that will dictate the number of cavities, etc. and, in turn, determine the size of the machine that will "fit around the mold." Often, however, this approach cannot be used, since the machine(s) to be used already exists and must be used. Obviously, the mold must physically fit between the tie-bars of the machine. The machine must also have adequate clamping pressure and a sufficient daylight opening for opening the mold and removing the part(s).

Plasticizing Capacity

Another consideration is the maximum amount of plastic required to fill the mold, including any runner system with a sprue that solidifies. The amount is usually 50% of machine shot capacity, or at most 60 to 70%, to ensure proper plasticizing action.

Economics

The economic decision could be the most important consideration in selecting the optimum number of cavities. For example, for

small volumes, anything more than a single-cavity mold is not economically justifiable. At the other extreme, very high volumes of product call for large numbers of cavities. Molds are normally built in multiples that result in a balanced cavity layout, 1, 2, 4, 8, 16, 32, 64, or 128 cavities being most usual. A general rule is that each time the number of cavities is doubled, the additional cavity cost is approximately 65% of the preceding cavity cost. Typically, the additional costs for a greater number of cavities must be balanced against manufacturing costs over a given length of time, three years being the average.

Here is a simplified guide formula that can help you determine the most economical number of cavities to put into various molds:

$$\text{Economic No. Cavities} = \sqrt{\frac{QRT}{ESC}}$$

where Q = total number of parts to be produced (1) for the life of the tooling or (2) for the market life of the plastic part, whichever is lowest
R = hourly rate of molding machine
T = cycle time (sec)
E = efficiency of molding machine, (percent) (normally 83%)
C = estimated cost per cavity
S = number of seconds per hour = 3,600

For example, a certain part has an estimated market of 480,000 parts/year for an estimated marketable life of 5 years. Cycle time is 45 sec. Machine burden or cost rate has a very low value of \$4.25/h. The estimated cost of each cavity is \$1,100. Then we have

$$\sqrt{\frac{2{,}400{,}000 \times \$4.25 \times 45}{0.83 \times 3{,}600 \times \$1{,}100}} = \sqrt{139.6} = 11.8$$

Cavity Draft

On most molded parts, there are features that must be cut into the surface of the mold perpendicular to the molding parting line. To properly release the part from the tool, such indentations almost always include a taper; the usual amount to start with is $\frac{1}{2}^\circ$. This is called a cavity draft, or draft in the direction of the mold.

The amount of draft required will depend on factors such as the type of plastic being processed, processing conditions, surface finish, etc. As an example, a highly polished surface will require less than an unpolished mold. Any surface texture will increase the needed draft at least 1° per side for every 0.001-in. (0.003-cm) depth of texture. Special mold cavity action can be used instead, as when the cavity side is moved slightly parallel to the wall direction when the part is to ejected. With elastomeric (rubbery) material, ejection may not require a draft (Fig. 4-8).

Cavity Packing

Plastic is a compressible fluid. Therefore it holds pressure and shrinks as it cools, requiring decisions to be made about the amount of overpacking necessary to minimize problems that occur when the plastic melt cools and shrinks, such as developing undue frozen stresses or causing flash. There is a tradeoff between overpacking and shrink, arrived at with a certain amount of guesswork based on experience. Computer programs are available that provide greater insight into the compressibility of plastic materials, so one is able to make better decisions.

Cavity Surface

The surface of the mold cavity reproduces its condition on a molded part. A significant advantage of the molding processes is the fact that different surface polishes and textures can be molded into the part. No secondary surface-finishing operations are required unless special finishes are required such as plating, hot stamping, etc. (Chap. 10). High-gloss, dull, matte, textured, etc. surfaces on parts (as well as combinations of them) are feasible.

Surface finish Tooling surfaces such as mold cavities require meeting certain surface finish requirements. Some times it is difficult

DNA Genotek Inc. Exhibit 2029 Page 66
Spectrum Solutions LLC v. DNA Genotek Inc. IPR2023-01424

to specify the finish required except with qualitative terms (dull, vapor-honed satin, shiny, etc.). In the past, standards such as the SPI (originally SPI–SPE) Mold Standard Finish (with six different finish surfaces) and ANSI Standard B46.1 Surface Texture (requiring extremely accurate surface measurements) were used. In view of the inadequacies of the old standards, SPI eventually issued a worldwide Mold Finish Standard. It has four distinct categories: A is the highest polish, having a diamond finish, B is a paper finish, C is a stone finish, and D is a blast finish. Each of the categories has three grades, with 1 being the best or highest and 3 the lowest. This standard lists only the final step in the mold benching: polishing. The mold surface must first be properly prepared removing machining marking, marks, etc.

Surface texturing Texturing means creating a pattern on a mold cavity surface that can be reproduced on the molded part. It can be done by cutting or by etching (chemical removal, or controlled corrosion). Tool texturing was initially used as a method to minimize the effect of flow lines, sink marks, and other flaws or functional needs on parts, but soon became a regular part of fabricated-part design. Since texturing can influence the type of cavity material used (type of steel, beryllium–copper, etc.), it is important in the initial stages of mold design to specify what texturing is to be done so no problems develop when the surface is to be treated.

Plaques are used to identify the different available surfaces. Tooling issues are numerous, such as steel type and hardness, surface finish requirements, metal removal for specified patterns, and mold cavity design. Because the process incorporates many hand-applied techniques, access to the surfaces that are to be decorated is crucial, and areas with restricted access should be discussed in the mold design stage. Adjustment of the cavity dimensions may be necessary to compensate for the metal removal that occurs during etching. Knife-edge inserts and cams are of particular concern.

The decorating options available are numerous and elaborate. As an example, there is microtexturing using a mechanical abrasion process, commonly referred to as sandblasting, where usually glass or aluminum oxide is impacted against the surface of the tool, leaving a lightly scarred steel surface. The appearances that are available are limited to matte textures and some stripes. However, their real value is the ability to reduce gloss levels. Graphic designs such as company logos and written or pictorial information may be etched into tooling surfaces where mechanical means may not be suitable.

Hobbing Hobbing is a technique in which a master model in hardened steel is used to sink a model shape into a heated mild steel mold cavity such as beryllium copper. This hob is larger than the finished plastic molded product. After hobbing, the metal cavity shrinks as it cools to the required size.

Clamping Forces

The clamping force required to keep the mold closed during injection must exceed the force given by the product of the live cavity pressure and the total projected area of all impressions and runners. The projected area can be defined as the area of the shadow cast by the molded part cavity when it is held under a light source, with the shadow falling on a plane surface parallel to the parting line.

With cold-runner systems for thermoplastics (or so-called hot-runner systems for thermosets), the projected areas of runners and sprue are included with the cavity(s). When hot-runner systems are used for thermoplastics (cold-runner for thermosets), the force to move the melt in the runner does not push apart the molds at the parting line; instead, it floats within the mold.

As an example, if the total projected area is 132 sq in. and a pressure of 5,000 psi is required in the cavity(s), based on the plastics being processed, the clamping force required is

Minimum clamping force
= projected area × plastic pressure cavity
= 132 sq in. × 5,000 psi = 660,000 lb

Table 4-6 Examples of clamping pressures required for PE, PP, and PS based on flow path length and section thickness

Average Component Section Thickness		Clamping Pressure Required (psi or kgf/cm^2 of Projected Area) at Ratio of Flow Path Length to Section Thickness				
in.	mm	200 : 1	150 : 1	125 : 1	100 : 1	50 : 1
0.04		—	9,960	9,000	7,200	4,500
	1.02	—	706	633	506	316
0.06		12,000	8,500	6,000	4,500	3,000
	1.52	844	598	422	316	211
0.08		9,000	6,000	4,500	3,800	2,500
	2.03	633	422	316	267	176
0.01		7,000	4,500	3,500	3,000	2,500
	2.54	492	316	246	211	176
0.12		5,000	4,000	3,100	3,000	2,500
	3.05	352	281	218	211	176
0.14		4,500	3,500	3,100	3,000	2,500
	3.56	316	246	218	211	176

or

$$= \frac{660{,}000\,\text{lb}}{2{,}000\,\text{lb}} = 330\ \text{tons}$$

Consider including a safety factor of about 10 to 20% to ensure sufficient clamping pressure, particularly when one is not familiar with the operation. Thus, the standard IMM maximum clamping force could be 330, 375, or 400 tons. A guide to the clamping pressure requirement for polyolefins and polystyrenes based on flow path length is given in Table 4-6.

For a true hydraulic fluid such as water, the clamping force required for each square inch of projected area would be equal to the unit pressure applied by the injection plunger. However, owing to the partial hardening of the plastic as it flows through the sprue and runners and into the cavity, the actual pressure exerted by the plastic within the cavity is much less than the applied plunger pressure. For this reason, an applied pressure of 20,000 psi would seldom require a clamping force of more than 5 or 6 tons/sq in. of projected area of the plastic shot (2,000 psi = 1 ton/sq in.).

For a given plunger pressure, the actual pressure developed within the cavity varies directly with the thickness of the molded section, and inversely with the melt viscosity. Thick sections require greater clamping force than thin sections because the plastic melt in a thick section stays semifluid for a longer time during the cavity-filling injection stroke. Similarly, a higher stock temperature, a hotter mold, larger gates, or faster injection will require a higher clamping pressure. As a general rule, good molding practice requires about 3 tons of clamp for each square inch of projected area of the molded shot.

By proper mold design and careful adjustment of molding conditions, it is sometimes possible to mold satisfactory parts with as little as 1 ton of clamp per square inch of projected area. However, it is unwise to attempt to operate a mold on this basis, as the range of permissible molding conditions would be seriously limited, and a long flow cavity fill could not be achieved.

It is also important to avoid applying too much clamping force to the mold. If a small mold is installed in a large machine and closed under full clamp, the mold can actually sink into the machine platens. Also, if the area of mold steel in contact at the parting line is insufficient, the mold may be crushed under the excessively high clamping force. Steel molds will begin to crush when the unit clamp pressure exceeds 10 tons/sq in. of contact area. In less severe cases, the mold components

may be distorted, or may fracture prematurely from fatigue.

Whereas the projected area determines the clamping force required, the weight or volume of the molded shot determines the capacity of the injection machine in which the mold must be operated. Note that the shot weight or shot volume includes the weight of the sprue and runners, except in hot-runner molds. Capacities of injection machines are commonly rated in ounces of polystyrene that can be injected by one full stroke of the injection plunger.

The calculated projection, on a plane perpendicular to the mold clamping force, of the total surface area of the moldings must not exceed the machine's maximum permissible molding area to be subjected to injection pressure. Machinery manufacturers usually provide this information.

Contact Area at Parting Line

Another item that requires attention is the contact area of the spacer blocks (7). The stress on these areas should be such as to prevent the embedding of these blocks into the plates, which would decrease the volume available for the ejection system. The safe tonnage that a mold base will take as far as spacer blocks are concerned can be calculated in this way. Let us take a $9\frac{7}{8} \times 11\frac{7}{8}$ in. (25.1 × 30.2-cm) standard mold base made of low-carbon steel. The weakest section of the spacer bar is at the clamping slot. For this size mold base, the width of the block is $1\frac{7}{16}$ in. and the width of the clamping slot $\frac{5}{8}$ in. The area will be $1\frac{7}{16}$ in. minus $\frac{5}{8}$ in., or $\frac{13}{16}$ in. × $11\frac{7}{8}$ in. × 2, since there are two blocks. Then we have:

Area × allowable stress = compressive force

The allowable stress for low-carbon steel is 25,000 psi. Thus,

$$\frac{13}{16} \text{ in.} \times 11\frac{7}{8} \text{ in.} \times 2 \times 25{,}000 \text{ psi} = 482{,}000 \text{ lb} = 241 \text{ tons}$$

Higher-strength steel throughout the base can double or even triple the ability to absorb the compressive force. The addition of supporting pillars will also increase the compressive force in proportion to the area that they provide. Thus, two 2-in.-diameter pillars would add the following force:

Area of a 2-in.-diameter pillar
= 3.14 sq. in. × 2 = 6.28 sq. in.
6.28 × 25,000 = 157,000 lb = 78.5 tons

The embedding problem may still arise, especially when changes are made in supporting bar dimensions or supporting blocks.

Sprue–Runner–Gate Systems

The sprue is the channel, cut in the stationary platen, that transports the melt from the plasticizing unit through to the runner system or direct through a sprue to the cavity (7, 103, 198, 348). The runner system basically transports the melt from the sprue to the gate and the gate transports the melt into the cavity. Complete systems of these melt flow systems are shown in Figs. 1-11, 4-10, 4-41, and 4-42. If there is no runner system and the plastic melt is transported directly from the sprue to the cavity, the sprue is known as a direct sprue gate.

The sprue, which forms the transition from the hot molten thermoplastic melt to the considerably cooler mold, is part of the flow length of the plastics and has to be of such dimension that the pressure drop is minimal and its ability to deliver material to the extreme "out" position is not impaired. The starting point for sprue size determination is the main runner, and the outlet of the sprue should not be smaller than the runner diameter at the meeting section. Thus, a $\frac{1}{4}$-in.-(0.64-cm)-diameter runner would call for a $\frac{7}{32}$-in.-(0.56-cm)-diameter "O" opening, for an average sprue length of 2 to 3 in. (5.1 to 7.6 cm). It has been established experimentally that for shots of 6 cu in. up to 20 cu in. (98.3 cu cm up to 327.7 cu cm), the $\frac{7}{32}$-in. (0.56-cm) O dimension will satisfy the need for a low pressure drop. For larger shots, a $\frac{9}{32}$-in. (0.71-cm) O opening would be indicated.

DNA Genotek Inc. Exhibit 2029 Page 69
Spectrum Solutions LLC v. DNA Genotek Inc. IPR2023-01424

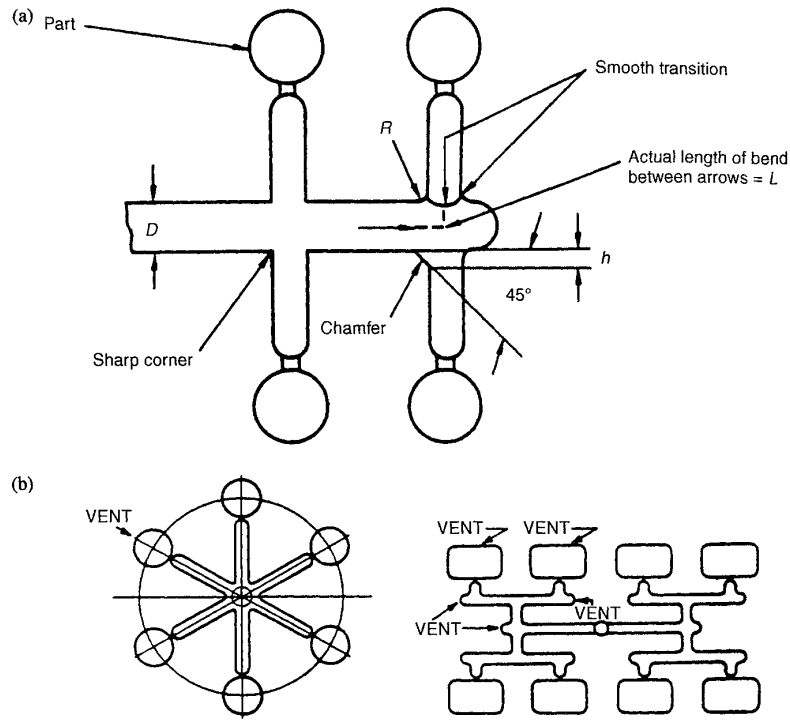


Fig. 4-42 (a) Effect of length of runner bends: example for ideal runner with $R = \frac{1}{3}D$ to $\frac{1}{2}D$. For sharp corners the effective length is $2.5L$; for a chamfer with $h = \frac{1}{3}D$ it is $2.5L$. (b) Balanced-spoke runner layout (left) and H-runner layout (right).

The material processing data give a range of runner sizes for each material. The smaller sizes can be applied for cases in which the length of runners does not exceed 2 in. (5.1 cm) and the volume of material is less than 15 cu in. (245.8 cu cm). For economic reasons, it is preferable to keep the runners on the smaller end, since that not only reduces the amount of regrind, but also accelerates the freezing of the gate, thus affecting cycle time. The pressure drop must be kept in mind. It becomes a matter of proportioning runners in relation to the spacing of cavities, wall thickness of parts, length of cavities, and corresponding gate sizes.

Basically, the distance from the injector (melt plasticator) of the injection machine to the mold cavity(s) should be as short as possible. However, different factors must be considered that could require longer distances. One factor, discussed earlier, is the number of cavities. Another factor relates to mold side actions that require longer runners. It is very important to allow sufficient space for cooling channels.

Perhaps the least-understood and least well applied factor is the inclusion of cooling channels for heat transfer from the plastic melt to the cooling liquid (for thermoplastics). Usually, insufficient space is allowed between cavities, particularly in molding the crystalline polymers (polyethylene, polypropylene, nylon, etc.) General information on cooling is reviewed later in this chapter.

Sprues

In single-cavity molds, the sprue usually enters directly into the cavity, in which case the sprue diameter at the point of cavity entry

should be approximately twice the thickness of the molded article at that point. Insufficient diameter of the sprue gate can cause excessive frictional heating and/or delamination of the plastic at the gate area, as well as wear of the metal.

Too large a sprue diameter requires a prolonged molding cycle, to allow the plastic sprue sufficient time to cool for removal. In all direct-sprue-gated cavities, an internal water fountain should be installed in the mold to cool the mold surface directly opposite the gate. All plastic injected into the mold impinges on this surface and causes a hot spot on the metal cavity wall.

In three-plate and hot-runner molds, the main sprue is designed as described above. The smaller sprues (also known as "subsprues"), which convey plastic from the runners to the cavities in such molds, are designed to converge toward the gates.

The sprue area has been the location of more than its share of problems in the injection molding process. The cause of most of these problems is the great temperature difference (about 300°F) between the nozzle and sprue. The nozzle is a transfer system and must maintain a temperature to keep the plastic in the liquid state, whereas the sprue is part of the mold-fill system and maintains a temperature conducive to solidifying the plastic.

The devices applied in the area of the sprue do not address a graduated temperature change between nozzle and sprue. Among the more frequent problems are nozzle freeze-off, materials degradation, and nonuniform melt. These problems are aggravated when the materials are highly crystalline or temperature-sensitive. The usual approach to solving sprue problems is to design tools that minimize the length and size of the sprue, use a heated sprue, or eliminate the sprue altogether.

Efforts to overcome the temperature difference between nozzle and sprue have concentrated on the nozzle, resulting in a variety of devices and modified types of nozzles (an example is shown in Fig. 4-43). When the fill difference is overcome by adding heat to the nozzle, severe problems can exist:

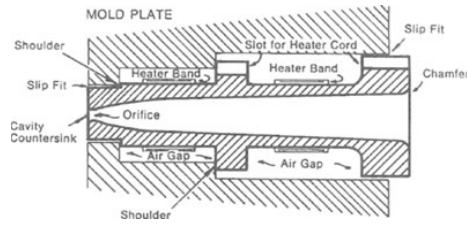


Fig. 4-43 Heated sprue bushing eliminates trimming and sprue scrap and reduces molding cycle for thermoplastics.

burned spots, knit lines, gas trapping, weakened parts, color change, streaking, black specks, blemishes, and increased scrap. The alternative of running with cooler temperatures leads to almost an equal generation of scrap, in this case related to cold spots in the melt. There are knit lines and surface blemishes, and, in addition, sticking sprues, plugged gates (especially using pin gates), and nozzle freeze-off. This situation tempts the operator to resort to crude on-the-spot remedies to keep production going. Among the more extreme have been cardboard insulators, long pieces of brass rod—even hammers and a torch.

To dispense with the sprue when using hot or insulated runner molds or to feed directly into the mold cavity, extended nozzles can be useful. They are suitable for single-impression work and, in the form of a manifold nozzle, for multiimpression work as well.

Sprue bushings provide an interface between the injection-machine nozzle and the runner system in the mold, and their design will vary greatly with the type of mold and injection machine required for a particular molding job. Sprue bushings are generally preengineered catalogue items, and it is usually a good idea to examine a large number of designs from various manufacturers before deciding on a bushing for a particular mold.

Runner Systems

Cavities should be placed so that (1) the runner is short and, if possible, free of bends, and (2) the supply of material to each cavity

is balanced. This means that the runners must be practically identical in both shape and size (length as well as the gate size). This becomes especially important for precision parts.

A balanced supply ensures that any change made in any one of the molding parameters will affect all cavities to the same extent. It is good practice to use a runner plate of the same grade of steel as the cavities, which has a surface machined to 50 rms (root mean square). In some applications, especially in cases of low usage of a mold, there is a tendency to machine the runner in the cavity plate instead. If a cavity protrudes on one side above the plate, a runner plate on that side is a must. Runner systems will vary in size and shape.

The surface finish of the runner system should be as good as that in the cavity, for example, machined to 50 rms. A good surface finish not only keeps the pressure drop low, but also prevents the tendency of the runner to stick to either half of the mold. Such sticking would aggravate the high stress in the area of the gate.

The runners in multicavity molds must be large enough to convey the plastic melt rapidly to the gates without excessive chilling by the relatively cool mold for thermoplastics. Runner cross sections that are too small require higher injection pressure and more time to fill the cavities. Large runners produce a better finish on the molded parts and minimize weld lines, flow lines, sink marks, and internal stresses. However, excessively large runners should be avoided, for the following reasons:

1. Large runners require longer to chill, thus prolonging the operating cycle.

2. The increased weight of a large runner system subtracts from the available machine capacity, in terms of not only the ounces per stroke that can be injected into the cavities, but also the plasticizing capacity of the heating cylinder in pounds per hour.

3. Large runners produce more scrap, which must be ground and reprocessed, resulting in higher operating cost and an increased possibility of contamination.

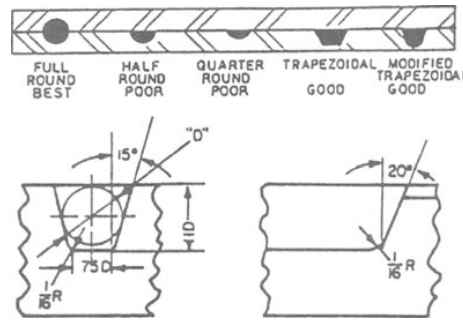


Fig. 4-44 Different shapes of runners.

4. In two-plate molds containing more than eight cavities, the projected area of the runner system adds significantly to the projected area of the cavities, thus reducing the effective clamping force available.

Note that these objections do not apply to hot-runner or runnerless molds.

Various shapes of runners are used (Fig. 4-44). A full round (i.e., circular cross section) runner is always preferred over any other cross-sectional shape, as it provides the minimum contact surface of the hot plastic with the cool mold. The layer of plastic in contact with the metal mold chills rapidly, so that only the material in the central core continues to flow rapidly. A full-round runner requires machining both halves of the mold, so the two semicircular portions are aligned when the mold is closed.

There are, however, many mold designs that make it desirable to incorporate the runner in one plate only. In that case, a trapezoidal cross section is used. If the trapezoid can be cut so that it would exactly accommodate a fully round runner of the desired diameter, and has sides tapered at 5 to 15° from vertical above the halfway line, that will be almost as good as the round runner.

Thermoplastic cold-runner systems Designing the smallest adequate runner system will maximize efficiency in both raw-materials use and energy consumption in molding. At the same time, runner size is constrained by the amount of pressure drop and injection capacity of the machine. Molders often seen unaware of the need to

balance these two equally important considerations.

Since molding a runner system does cost money, it makes sense to minimize the amount of nonsalable material molded into the runner. Even though the runner system will probably be reground and recycled, it is still important to keep its weight and size to an absolute minimum because some plastics tend to degrade during repetitive processing. A properly designed runner will help not only reduce costs, but also preserve part quality.

Traditionally, there have been a number of misconceptions about proper runner design, many of which are still prevalent in molding shops. In the past, many injection molders and tool builders felt that the larger the runner, the faster the melt would be conveyed to the cavity. They also believed that the lowest possible pressure loss through the runner system to the cavity would be the most desirable. Runners were commonly machined into the mold with these objectives in mind. However, it is, in fact, important to select the minimum runner size that will adequately do the job with the material being used.

Consider two runner systems designed for nylon, for example. A traditional runner might weigh 50 g, whereas a well-planned, smaller yet adequate runner would weigh (say) 20 g. Assume the mold produces 750,000 shots/year. At an electrical cost of 5¢/kWh and energy requirement of 350 Btu/lb to plasticate nylon, the cost of molding the extra material in the overweight runner system is about $300/year. The latter figure assumes close to 100% mechanical and electrical efficiency. Given the actual efficiency factors typical of molding machines, however, an added cost of $1,000 per mold per year with a poorly designed runner is not unlikely. Multiply this amount by the number of machines in your shop, and you will have an idea of how much energy and money can be wasted by not carefully considering runner size.

Although properly sizing a runner to a given part and mold layout is a relatively simple task, it is often overlooked because the basic principles are not widely understood. For one thing, few processors are comfortable with using the straightforward arithmetical calculations involved. Also, the rules of runner design can be easily neglected in the rush to commit a part design to the toolmaker. Lack of familiarity with the rules of optimum runner design undoubtedly leads processors to think there is some mystery involved, which is not the case.

There are techniques for computing the minimum runner size required to convey melt at the proper rate and pressure loss to achieve optimum molded part quality. As a result, runner design has evolved from pure guesswork into an engineering discipline based on fundamental plastic flow principles. The molder who neglects the opportunity to engineer his or her runner systems is likely to miss a major opportunity to lower costs and improve productivity.

The computations are based on a key rheological property of the material to be molded. This property is the material's shear rate vs. its melt viscosity at several commonly encountered melt temperatures for the material. Usually, this information is available from your resin supplier, and it is frequently displayed in molding manuals for individual materials. Figure 4-45 provides an example of such data.

Since no single calculation will do the job, it is necessary to start with a reasonable runner size, estimated on the basis of prior experience, that can then be refined with the aid of calculations. Initial considerations include the part weight and configuration and its performance or appearance requirements. For example, it is desirable when molding nylon to fill the part within 2 to 3 sec. In fact, the same is true of the majority of injection-molded parts made from crystalline thermoplastics, though not necessarily for amorphous resins.

Engineering a runner system requires an understanding of the pressure drop of the plastic as it passes through a channel. This pressure drop is controlled primarily by the volumetric flow rate or injection speed, melt viscosity, and channel dimensions. Although it is possible to reduce the melt viscosity by increasing the melt temperature—hence reducing the pressure drop—most injection molding materials have an "ideal" melt

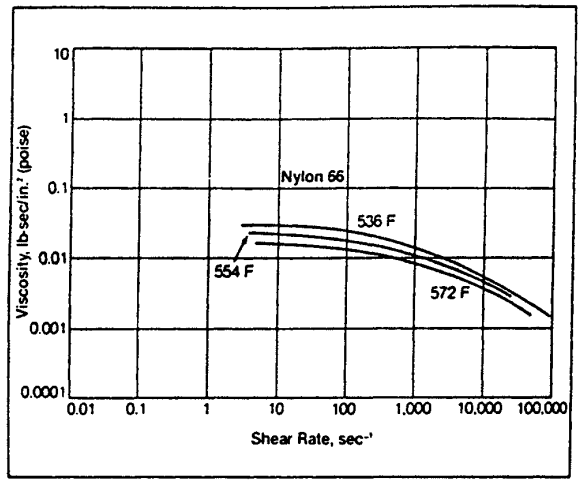


Fig. 4-45 Viscosity curve, typical of those available from most plastic material suppliers. Such curves can also be determined by the user with proper equipment; see Chap. 12. This information is essential to calculating the optimum runner diameter.

temperature that provides fast cycles and optimum part quality. Thus, runner engineering should start by assuming an ideal melt temperature. This temperature can be found in the resin supplier's molding manual.

The other assumption that must be made initially is the amount of pressure drop that can be tolerated. The IMM is usually capable of delivering 20,000 psi (138 MPa) of pressure. Since common sense forbids designing a mold to demand the absolute pressure limit of the machine, the mold should be designed so that the pressure required is somewhat less that the machine's capacity. A good value to assume is 10,000 to 15,000 psi (69 to 103 MPa). For the example shown here, a 15,000-psi injection pressure is assumed.

Unless the part design is unusual—such as long, thin parts—or experience dictates otherwise, a pressure of 5,000 psi (34 MPa) is usually adequate to fill and pack out most parts. This means, in our example, that the runner system can be designed for a 10,000-psi pressure drop. How is this done? The starting point is our hypothetical eight-cavity, balanced-runner layout, shown in Fig. 4-46. We assume that all runners are the full-round type, material specific gravity is 1.0, and part weight is 15 g. For eight cavities together, the total amounts to 120 g or 7.31 cu in. (120 cu cm). Lengths of the primary, secondary, and tertiary runners are shown in the figure. We also assume a typical fill or injection time of 3 sec. The foregoing are all fixed parameters; what remains to be determined is the optimum runner diameter. To start with, we estimate the diameters as shown, going by prior experience and typical industry practice.

Runner volume V is calculated as follows:

$$V = \pi r^2 L$$

where r = runner radius
L = length

Thus,

$$\text{Primary runner: } V_p = \pi(0.125)^2(10) = 0.49 \text{ cu in.}$$

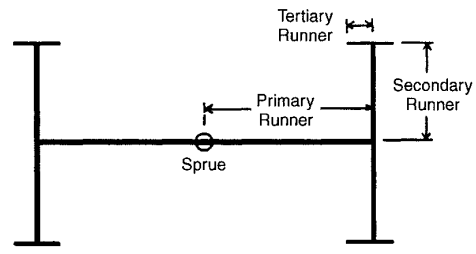


Fig. 4-46 Example of 8-cavity mold runner system.

DNA Genotek Inc. Exhibit 2029 Page 74
Spectrum Solutions LLC v. DNA Genotek Inc. IPR2023-01424

Secondary runner: $V_s = \pi(0.100)^2(12)$
$= 0.38$ cu in.
Tertiary runner: $V_t = \pi(0.075)^2(8)$
$= 0.14$ cu in.
Total shot volume (runner + parts)
$= 7.31 + 0.49 + 0.38 + 0.14$
$= 8.32$ cu in. (136.3 cu cm)

Since the flow splits at the intersection of the sprue and primary runner into two identical halves of the runner system, we need only calculate the pressure loss through one half of the mold. The volume of melt that must be conducted through the primary runner in this half of the system is 4.16 cu in (68.2 cu cm). Given our specified 3-sec fill time, the desired flow rate is 1.39 cu in./sec (22.8 cu cm/sec). This is the volumetric flow rate Q.

Now the shear rate S_r can be calculated

$$S_r = \frac{4Q}{\pi r^3} = \frac{4(1.39)}{\pi(0.125)^3} = 906\,\text{sec}^{-1}$$

The melt viscosity at this shear rate and the specified melt temperature must be read from a chart similar to Fig. 4-31. For this hypothetical example, the apparent melt viscosity is $\mu = 0.016$ lb-sec/in. (poise).

Next, we calculate the shear stress S_s

$$S_s = \mu S_r = (0.016)(906) = 14.5 \text{ psi}$$

Finally, the pressure drop P through that runner segment is calculated:

$$P = \frac{S_s(2L)}{r} = \frac{14.5(2)(5)}{0.125} = 1{,}160 \text{ psi}$$

Now the next runner segment must be considered. The total volumetric flow through each secondary runner is 4.16 cu in. minus the volume in the primary runner, so the runner flow after it is

$$\frac{4.16 - 0.25}{2} = 1.95 \text{ cu in.}$$

(Remember that the flow splits in half again at the secondary runner.) The volumetric flow rate in each secondary runner segment is 1.95/3 or 0.65 cu in./sec. Thus,

$$S_r = \frac{4(0.65)}{\pi(0.100)^3} = 827\,\text{sec}^{-1}$$

The melt viscosity at the shear rate is 0.017 poise. Therefore,

$$S_s = (0.017)(827) = 14.0$$
$$P = \frac{(14)(2)(3)}{0.100} = 840 \text{ psi}$$

The volumetric flow through each tertiary runner can be calculated by subtracting the volumes of primary and secondary runners, or simply by adding together the total tertiary runner volume and total part volume and dividing by eight cavities:

$$\frac{0.14 + 7.31}{8} = 0.93 \text{ cu in.} \quad (15.24 \text{ cu cm})$$

The volumetric flow rate is thus 0.93/3 or 0.31 cu in./sec, and

$$S_r = \frac{4(0.31)}{\pi(0.075)^3} = 936\,\text{sec}^{-1}$$

The viscosity corresponding to this shear rate is 0.016 poise, and

$$S_s = (0.016)(936) = 15.0$$
$$P = \frac{(15)(2)(1)}{0.075} = 400 \text{ psi} \quad (2.76 \text{ MPa})$$

The total pressure loss from the sprue to each gate is the sum of the pressure losses through each segment:

Pressure loss (total) $= 1{,}160 + 840 + 400$
$= 2{,}400$ psi (16.54 MPa)

This preliminary calculation shows that much smaller channels can be designed to accommodate a 10,000-psi (68.9-MPa) pressure loss. By repeating the calculations for progressively smaller runner diameters until we reach the targeted pressure loss, we eventually obtain the assumed runner diameters shown in Fig. 4-46.

In calculating and recalculating optimum runner diameters, the question may arise as to what is the appropriate relationship between the diameters of primary, secondary, and tertiary runners. In fact, there is no hard and fast rule for this, and the choice is somewhat arbitrary. It is logical, however, that since each successive stage of the runner system carries less melt than the previous stage, the successive runner diameters normally run smaller.

At times, it is necessary to build molds where the number of cavities is not two, or

DNA Genotek Inc. Exhibit 2029 Page 75
Spectrum Solutions LLC v. DNA Genotek Inc. IPR2023-01424

it is not possible to balance the cavity layout for equal flow distances to all cavities. Although this type of design presents no particular problem in molding parts with loose tolerances, the effect on dimensions and part quality must be considered carefully when designing runner systems for critical parts. The primary objective in the latter case is to design a runner system so that all cavities fill at the same rate. This is necessary to ensure that they cool at the same rate and provide uniform shrinkage; surface gloss can also be affected. Molders will frequently try to balance the fill rates of individual cavities by changing the gate size. While this has some utility, it is a relatively ineffective way of making up for unbalanced runner layouts. The land length of the gate is too short to make any significant difference in pressure drop from one cavity to another. It is much better to vary the runner diameters and control fill rate.

Figure 4-47 shows an actual six-cavity mold that was used to make a large automotive part, in which the sprue was offset from the center of the runner system. Since we want all the cavities to fill at the same rate, what is required is a computation of the runner diameters that will provide the same pressure drop from the sprue bushing to the gate of each cavity. Clearly, since the runner lengths are different for each pair of cavities, different runner diameters will be required as well. As shown by a previous equation, pressure drop is proportional to runner length, so it is evident that the longer runner segments will need to be slightly wider. Figure 4-47 shows the actual lengths and diameters for each segment of the runner system. Note that the total pressure drops into the various cavities are similar though not identical; it is often impractical (and unnecessary) to exactly balance the pressure drop into each cavity. In this case, it was considered impractical to go smaller than $\frac{1}{8}$ in. for the diameter of the secondary runners closest to the sprue in order to raise the pressure drop there to a level closer to that of the other secondary runners. In actuality, the parts all filled uniformly, despite some degree of disparity in the pressure drop leading into the cavities.

Figure 4-48 illustrates an extreme case of how runner diameter, not gate size, can be used to balance flow and pressure drop in an unbalanced cavity layout. Here again, we have an actual 10-cavity family mold, which produced dissimilar parts ranging in size from 2 in. (5.1 cm) in diameter by 1 in. (2.54 cm) long to $\frac{1}{4}$ in. (0.64 cm) in diameter by $\frac{1}{2}$ in. (1.27 cm) long. Nonetheless, as the numbers in the drawing show, it was possible to balance the pressure drops into the cavities quite closely.

The principles used in calculating the optimum diameter of the final runner segments of a three-plate mold with multiple drops into the cavity are the same as those discussed above. However, for most three-plate molds with multiple drops, it is frequently difficult to design them so that an equal volume of melt passes through each drop. For circular parts with tight tolerances, it is nonetheless highly desirable that the part fill equally from each gate in order to minimize out-of-roundness. The answer is to use the procedures already described to calculate the pressure loss through each drop and size the runner drop accordingly. Since the drops are usually tapered, the diameter is not constant. The difficulty can be circumvented by using the diameter at half the length as a basis for this calculation.

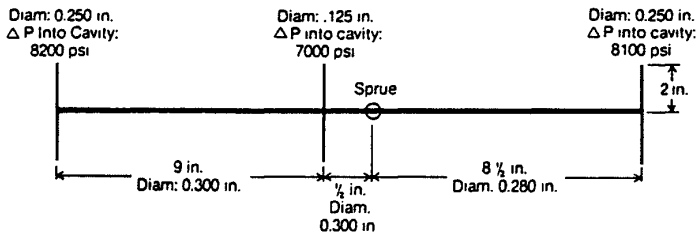


Fig. 4-47 Example of 6-cavity mold runner system.

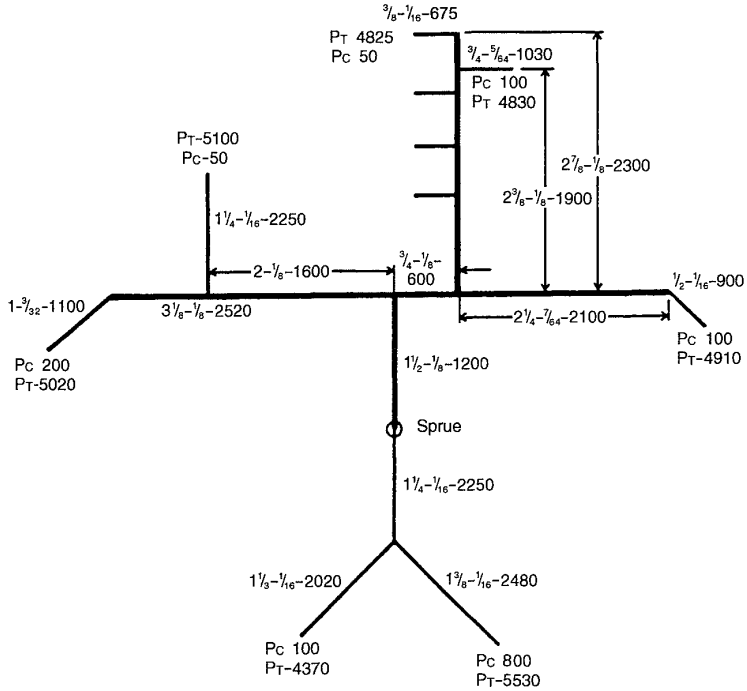


Fig. 4-48 Example of 10-cavity mold runner system for an automotive part (P_c = pressure drop in cavity and P_t = total pressure drop).

Sucker pins in the drop area will obviously influence the pressure loss and can provide additional restrictions to help equalize flow into each drop. Both the length and diameter of the sucker pin can be used to regulate the flow. However, it is seldom necessary to calculate the pressure loss across a sucker pin exactly; a reasonable assumption will usually prove adequate.

For those who cannot go through the calculations, industry-recommended runner diameters for different plastics are provided in Table 4-7.

Thermoplastic hot-runner systems There is nothing new about the runnerless molding process. Tools for this type of molding have been in use since the 1940s, with most of the activity starting during the early 1960s. Yet because of certain problems these molds have encountered (drooling, freeze-off, leakage, high maintenance, and others), runnerless molding has been used with some irregularity. However, new design concepts and tool-building methods have overcome these

Table 4-7 Recommended TP cold-runner diameters for use if runner size is not calculated

Material	Diameter	
	in.	mm
ABS, SAN	0.187–0.375	4.7–9.5
Acetal	0.125–0.375	3.1–9.5
Acrylic	0.312–0.375	7.5–9.5
Cellulosics	0.187–0.375	4.7–9.5
Ionomer	0.093–0.375	2.3–9.5
Nylon	0.062–0.375	1.5–9.5
Polycarbonate	0.187–0.375	4.7–9.5
Polyester	0.187–0.375	4.7–9.5
Polyethylene	0.062–0.375	1.5–9.5
Polypropylene	0.187–0.375	4.7–9.5
PPO	0.250–0.375	6.3–9.5
Polysulfone	0.250–0.375	6.3–9.5
Polystyrene	0.125–0.375	3.1–9.5
PVC	0.125–0.375	3.1–9.5

DNA Genotek Inc. Exhibit 2029 Page 77
Spectrum Solutions LLC v. DNA Genotek Inc. IPR2023-01424

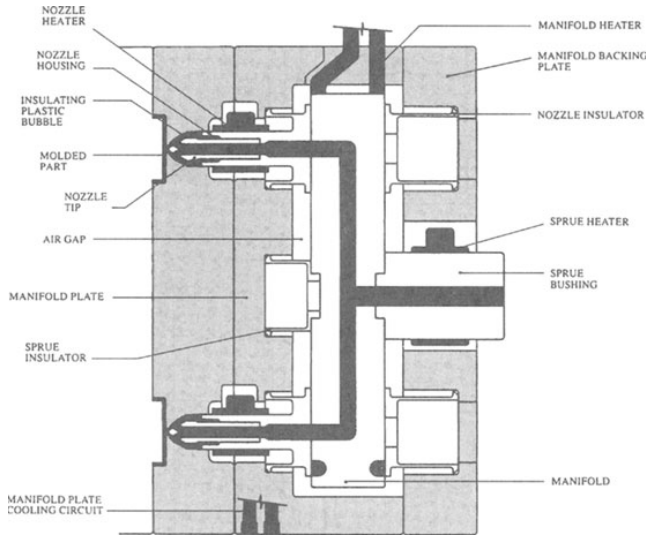


Fig. 4-49 Example of cartridge-heated hot-runner system with terminology.

objections, and today's tools for runnerless molding are highly efficient and relatively fault-free.

The term "runnerless" refers to the fact that the runner system in the mold maintains the plastic resin in a molten state. This material does not cool and solidify, as in a conventional two- or three-plate mold, and is not ejected with the molded part. It is a logical choice for any high-speed operation in which scrap cannot be reused.

There are two design approaches for tools used in runnerless molding: the insulated runner and hot runner. Insulated-runner molds have oversize passages formed in the mold plate. The passages are of sufficient size that, under conditions of operation, the insulating effect of the plastic combined with the heat applied with each shot maintains an open flow path. Runner insulation is provided by a layer of chilled plastic that forms on the runner wall.

Hot-runner molds, which are the more popular of the two types, are generally built in two styles. The first is characterized by internally heated flow passages, the heat furnished by a probe or torpedo located in the passages. This system takes advantage of the insulating qualities of the plastics to avoid heat transfer to the rest of the mold.

The second, more popular system consists of a cartridge-heated manifold with interior flow passages. The manifold is designed with various insulating features to separate it from the rest of the mold, thus preventing heat transfer (Figs. 4-49 and 4-50).

Of the two basic systems, the insulated runner has seen less attention in recent years. Although the insulated-runner molds are generally less complicated in design and less costly to build than hot runners, they also have a number of limitations, including freeze-up at the gates, fast cycles required to maintain the melt state, long startup periods to stabilize melt temperature and flow, and problems in uniform mold filling. The predominant style of hot runners in industry today is the externally heated manifold type.

A great deal of interest has centered on hot-runner molds since the plastics industry improved the distribution of heat and level of temperature control. Furthermore, the industry has developed numerous components that enhance the design and construction of hot-runner molds. These standard components include a variety of cartridge-, band-, or coil-heated machine nozzles, sprue bushings (Fig. 4-51), manifolds, and probes; heat pipes; gate shutoff devices; and electronic controllers for various heating elements. Because

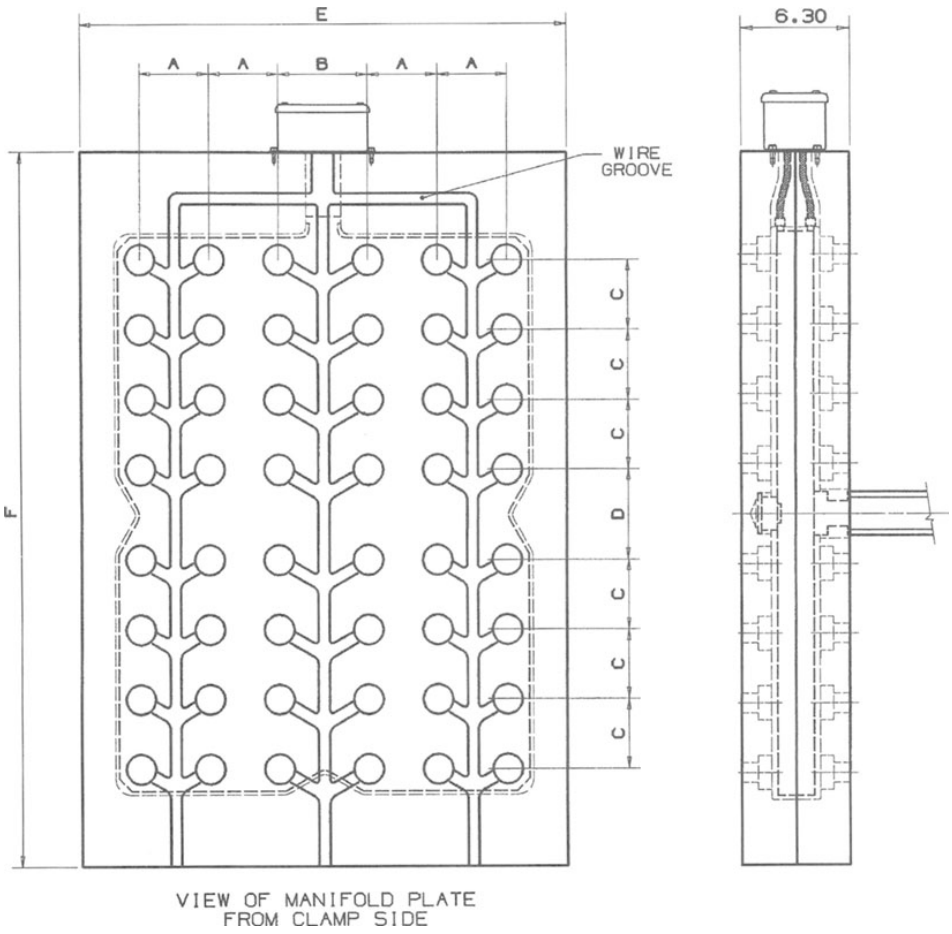


Fig. 4-50 Example of a hot manifold used in a stack mold that delivers melt to 48 cavities on each side (total 96 cavities).

of this interest, the remainder of this section will focus on hot-runner molds.

The design of hot-runner molds should take into account the thermal expansion of various mold components; this applies mainly to the center distances between the nozzles, supports, set bolts, and centering points. The bends in the hot runners to the nozzles should be generously radiused to prevent dead corners. In the design, each nozzle contains a capillary to act as a valve to prevent plastic leakage. Heating elements positioned around the nozzles provide proper temperature control. When thick-walled articles are molded, the long after-pressure time may necessitate the use of nozzles with needle valves, as capillaries tend to freeze up rather quickly.

Heater loading in hot-runner manifolds is:

1. For general-purpose materials (polystyrene, polyolefins, etc.)

15 to 20 W/cu in. of manifold
(0.92 to 1.22 W/cu cm)

2. For high-temperature thermoplastics (nylon, etc.)

20 to 30 W/cu in. of manifold
(1.22 to 1.83 W/cu cm)

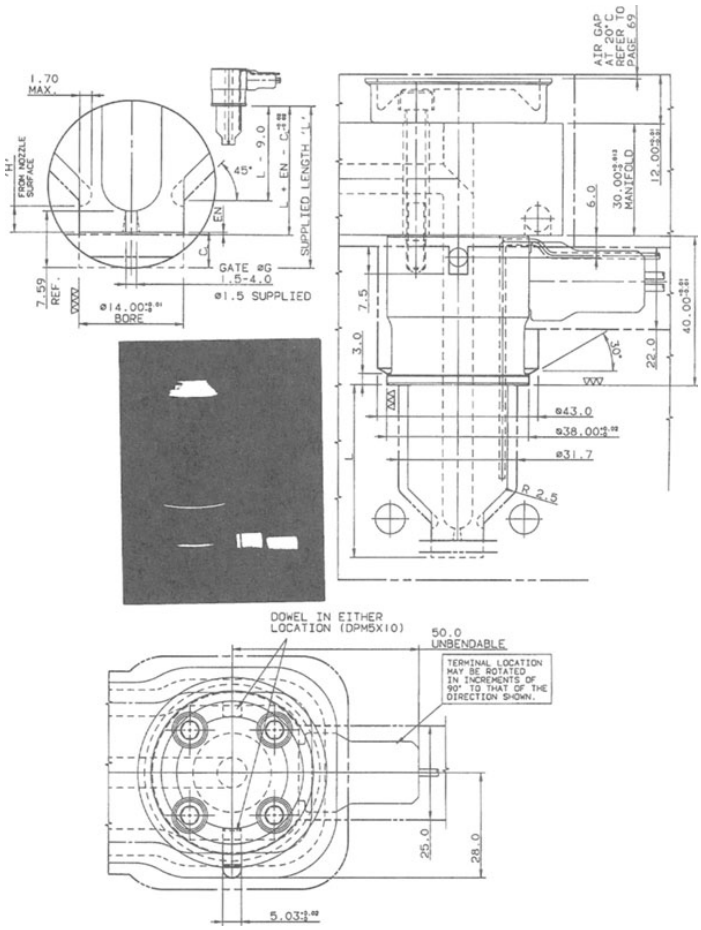


Fig. 4-51 Example of Mold Masters hot sprue.

Heater loading in the gate torpedo for insulated runner molds is 35 W.

Advantages and disadvantages A major advantage of hot runners (for thermoplastics) is that they reduce or eliminate scrap. Unlike cold-runner systems in which plastic solidifies in the runner and is ejected with the part, plastic remains melted in the heated runner, ready for the next injection cycle. A major portion of the cycle time for a plastic part is cooling time, which is the amount of time it takes the plastic to set prior to mold open and ejection. In a cold-runner mold, the thickest wall section is often found in the cold runner, and the molding cycle may wait until the runner is solid enough to be ejected. Whether it is freefall or by sprue picker, the elimination of the runner results in a reduction in the cooling portion of the cycle, thus reducing the overall cycle time. Cycle time can be reduced by as much as 50%.

The elimination of the cold runner means less recovery time is required, since the injection unit does not have to plasticate the cold runner. If the runner made up 30% of the shot weight, this would reduce the recovery time proportionally. If recovery time hindered the overall cycle previously, this would also reduce cycle time.

The reduction of the overall shot weight also means that injection time is reduced,

since the same injection rate needs to be maintained for required fill rates. Also, the resin's flow path is much shorter.

The elimination of the runner-plate movement reduces the clamp motion, since the stroke is shortened and runner stripper plates controlled by shoulder bolts are not required. With shoulder-bolt ejection, the stroke needs to be profiled to ensure that the shock loading is controlled. Elimination of this action allows full clamp speed to be incorporated, again reducing cycle time.

Mold-open dwell time is reduced, since the system does not have to wait for the ejection of the runner, further reducing cycle time. The elimination of the cold runner reduces the amount of plasticating required by the injection unit, which in turn reduces the energy consumed per part. The hot-runner approach eliminates the need for a sprue picker and grinder, which also require energy and personnel to operate.

A reduction in shot size and elimination of the runner mean a shorter injection stroke and less pressure is needed to fill the mold, all adding up to additional energy savings. The reduction in pressure loss during fill is achieved with the use of heated flow channels.

As the resin flows through the cold runner, a solid layer sets up on the channel wall, restricts flow, and requires greater injection pressures from the machine to help overcome losses. The higher pressure at the injection end of the runner is required to achieve the needed pressure to overcome the gate restriction, flow losses, and cavity filling. Keeping the resin molten in the hot runner reduces the pressure drop to each cavity, since the flow is less obstructed.

The flow length found in a hot-runner system also tends to be shorter, further reducing the pressure losses found in a cold-runner system. Reductions of peak injection pressure from 1,250 to 700 psi (8.6 to 4.8 MPa) oil pressure have been realized.

The hot-runner system provides a balanced flow to each cavity, resulting in consistent part weight from cavity to cavity. Balanced flow also produces fewer rejects.

Reduced injection pressure means less stress in the part, providing better structural quality. A reduction in pressure results in easier filling of the cavities, which reduces the deflection in both the platens and mold, reducing the amount of flash, again improving quality.

Although we tout the benefits of hot-runner technology and recognize that nothing on earth is perfect [see one definition of perfect in Reference 6], it is important to understand that the technology increases the cost of a mold and the extra expense needs to be justified by the application. On average, a hot-runner system adds 10 to 15% to a mold's cost, but sometimes it could double the mold's cost.

Such higher cost can best be justified for high-volume production, the molding of expensive plastics, and high-quality molding where gate vestige should be minimal. Parts made with hot-runner systems can weigh less than 1 g or as much as 160 kg (350 lb) and can have extremely large volumes (e.g., like a big trash container). As engineering plastics becomes more sophisticated and expensive, there will be more of a need for hot-runner systems to eliminate or significantly reduce the waste of plastics or build up their residence time.

Retrofits Molds using cold-runner technology offer opportunities to improve profitability with hot runners. If a conversion to hot runners provided cycle savings of only 10% for a 40-machine plant, this would free up four machines, or it could increase the revenue from the plant by 10% without adding any new machines. The elimination of a cold runner, as previously mentioned, can also reduce energy consumption and mold maintenance, eliminate granulator and sprue picker, and improve part quality and the efficiency of cavities.

In some cases, complete conversions from cold- to hot-runner systems are precluded by existing mold design. However, a combination hot–cold runner could be implemented, providing many of the same advantages.

The hot-runner conversion can be made on both two- and three-plate cold-runner molds. The conversion can be either to a full hot runner or a hot–cold combination. The latter

DNA Genotek Inc. Exhibit 2029 Page 81
Spectrum Solutions LLC v. DNA Genotek Inc. IPR2023-01424

would have a hot runner feed a smaller cold runner, providing many of the benefits of hot runners.

The degree of conversion can only be determined after the existing mold design is reviewed. This helps to ensure that a hot-runner conversion is viable and determine what modifications need to be made. In some cases, the complexity of the mold or part may not allow direct gating with a hot runner. This situation may require an approach that employs a hot–cold runner system.

A hot–cold runner system is one in which a hot runner feeding a cold runner, which in turn feeds the cavities. This approach substantially reduces the runner weight and can provide a more balanced delivery of resin. The elimination of the sprue and thick feed runners offers the advantages of smaller shot size, reduced injection pressure, and possible cycle savings.

A hot–cold combination may also require sucker pins and sucker-pin motion to eject the runner. This can be determined after the mold design is reviewed. The following should be weighed when you consider a conversion:

- *Cavity material.* The existing cavity may need to be modified to accommodate the hot-runner nozzle tip. The existing material may not be reworkable; new cavities or gate inserts may be required.
- *Gating style.* The gating required by the part needs to be reviewed to ensure it can be accommodated. The existing cavity must provide space to install a hot-runner probe. The location of the gate may need to be changed if insufficient space or cooling exists. The type of resin will also be a factor in the gating style, as some are more degradable than others.
- *Gate cooling.* The addition of the hot tip into the cavity requires a close look at the cooling in and around the gate to ensure that the desired thermal equilibrium can be achieved to produce consistent-quality gates.
- *Shut height.* The hot-runner system may add to the shut height of the mold. This needs to be considered along with conversion constraints.
- *Plate movement.* Many two- and three-plate molds use stripper bolts to generate the ejection force and plate motion during clamp open. The conversion may eliminate the need for this by using the machine ejector plate.
- *Machine sequence.* The change from a cold runner to a hot runner eliminates the cold sprue. The operating sequence on many existing injection molding machines is to inject, hold, recover, and then decompress. Recovering with back pressure keeps the resin in the manifold under pressure. The screw decompressing afterward tends to decompress the resin in the barrel, not that in the hot runner. This type of sequencing may cause a variation in gate quality.

Computer-aided designs There are different ways of designing hot-runner systems. Hot-runner manifold systems are divided into externally heated and internally heated systems on the basis of their method of design. Expanding on this previously reviewed subject, we note that internally heated systems have melt flowing over or along the heated mandrel. The dimensions of the melt channel in this case generally cannot be clearly defined, since the width of the gap in the ring channel depends on the thermodynamic boundary conditions. In externally heated systems, the melt flows through a tube to the individual hot-channel nozzles (Fig. 4-52). Since the runner dimensions are precisely defined, the pressure loss in an externally heated system can be easily calculated using an appropriate CAD software program. An example is that developed by the Plastics Technology Group at U-GH Paderborn in cooperation with Gunther Heibkanaltechnik GmbH Frankenberg/Eder, Germany (7).

Recognize that there is a distinction between naturally balanced and unbalanced hot-runner systems. A naturally balanced hot-runner manifold is characterized by flow channels of the same geometry (channel lengths, diameters) and, consequently, the same rate of melt flow from each of the nozzles. In an unbalanced system, the flow lengths to the nozzles are different, and they

DNA Genotek Inc. Exhibit 2029 Page 82
Spectrum Solutions LLC v. DNA Genotek Inc. IPR2023-01424

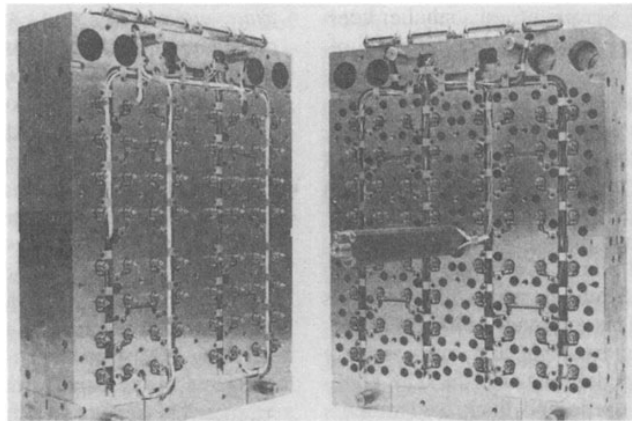


Fig. 4-52 Husky 96-cavity hot-runner mold manufactured via CAD/CAM/CAE and used in a stacked mold system.

can have different diameters. The following points have to be taken into account in the rheological design of a runner system:

- The pressure loss in the runner system must be as low as possible.
- So as to avoid dwell-time problems like plate-out, it is advisable that a particular limiting shear rate not be exceeded. The relevant limiting values for various materials are arrived at by experience.
- In systematic design, the channel diameters have upper (plate-out problems) and lower (pressure loss too great) limits. For this reason, the diameter that is specified cannot always be the one that is best rheologically.
- The hot-runner system should be built in the most systematic way possible and also be usable in different molds (development of modular systems).

In this connection, it should be mentioned that the thermal and mechanical layout also must be built into the systematic overview.

Each hot-runner system can, in principle, be designed so that the lengths of all flow channels to a set of cavities are the same. Because the flow lengths are necessarily long, there is certainly a large loss of pressure in the hot-runner system. To reduce pressure loss, the best policy is to specify large diameters and short flow lengths to individual injection points. Such a design procedure results in an unsymmetrical system. In order to balance it for a particular operating point, one possibility is to adjust the channel diameters so that, at the operating point, the manifold behaves like a balanced system with small pressure loss.

However, determination of the corresponding channel diameters takes quite some time, since flow impedances over the various flow lengths have to be calculated. This effort can be reduced by means of a dedicated computer program for the calculation of pressure loss and balancing of hot-runner systems. The program developed by U-GH Paderborn provides answers to questions such as the following:

- What does the volume-flow distribution of an unbalanced system look like?
- How much pressure loss is there in the runner system, and where do the greatest pressure losses occur?
- How must the channel diameters of an unbalanced system be modified to provide a balanced system at the operating point?
- How does a balanced system behave if some of the cavities are defective and the corresponding nozzles blocked?
- How does a balanced system behave if the operating point is changed (injection rate, melt temperature, material)?

Not all materials or all parts are equally adaptable to runnerless molding, so each case

DNA Genotek Inc. Exhibit 2029 Page 83
Spectrum Solutions LLC v. DNA Genotek Inc. IPR2023-01424

must be judged individually. Here is a checklist of considerations:

1. *Material.* Has it been processed by runnerless molding before? What does the materials supplier recommend? Not all of the thermoplastics have been molded via runnerless techniques, and the major problems are encountered with heat-sensitive materials, in which the time–temperature relationship can be a problem. However, with today's technology, even the acrylonitriles and polyethylene terephthalate are being run successfully on hot-runner molds.

2. *Part.* Is the part weight sufficient? With current technology, a very small part may not require sufficient material to be purged through the nozzle tip, and degradation may occur from excessive residence time in the heated channel. Does the part require a runner? For instance, in the case of a family mold, it might be desirable to leave the parts together on a runner system until they reach the assembly station.

3. *Process.* Is the viscosity of the material (nylon, e.g.) such that a positive, drool-free shutoff is required?

4. *Volume.* Does the run justify the additional expense of a hot-runner system? Although there is no firm figure on how much more runnerless molding will cost than cold-runner molds, the tooling cost could run 5 to 7% more for standard tooling and applications and substantially more for nonstandard tooling. The additional mold cost must be compared with the anticipated savings in machine hours, scrap, etc.

To clarify a point, the term "runnerless mold" is a misnomer. With the exception of a mold with a single cavity that is fed directly from the machine nozzle, all injection molds have a runner system. This term originated in the use of insulated or heated runner channels in which the resin does not cool and solidify. No plastic is ejected from the runner channel when the mold is opened and the mold part ejected. Thus, the term runnerless is indicative of the absence of scrap from the runner system; a more accurate expression would be "runnerless molding."

Gates

The gate is given a smaller cross section than the runner so that the molding can be easily degated (separated from the runners). The positioning and dimensioning of gates are critical, and sometimes the gates must be modified after initial trials with the mold. Feeding into the center of one side of a long narrow molding almost always results in distortion, the molding being distorted concave to the feed. In a multicavity mold, sometimes the cavities closest to the sprue fill first and the farther cavities later in the cycle. This condition can result in sink marks or shorts in the outer cavities. It is corrected by increasing the size of some gates so that the simultaneous filling of all cavities will result.

The location of the gate must be given careful consideration, if the required properties and appearance of the molding are to be achieved. In addition, the location of the gate affects mold construction. The gate must be located in such a way that rapid and uniform mold filling is ensured. In principle, the gate will be located at the thickest part of the molding, preferably at a spot where the function and appearance of the molding are not impaired. In this respect, it should be noted that large-diameter gates require mechanical degating after ejection and always leave a mark on the product. It is for this reason that in small or shallow moldings, the gate is sometimes located on the inside. However, this necessitates mold release from the direction of the stationary mold half, which interferes with effective cooling and generally increases mold cost.

Furthermore, the location of the gate must be such that weld lines are avoided. Weld lines reduce the strength and spoil the appearance of the molding, particularly in the case of glass-fiber-reinforced plastics.

Also, the gate must be so located that the air present in the mold cavity can escape during injection. If this requirement is not fulfilled, either short or burnt spots on the molding will be the result.

During the mold filling, thermoplastics show a certain degree of molecular orientation in the flow direction of the melt (as

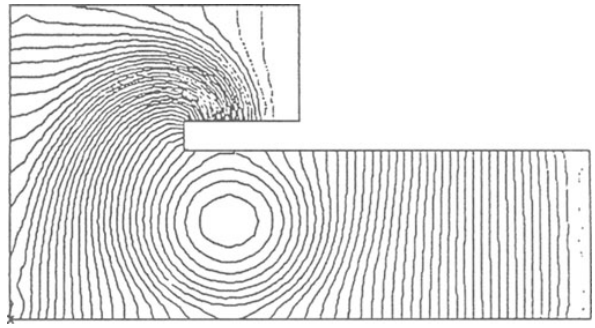


Fig. 4-53 Single-gate flow pattern.

previously reviewed), which affects the properties of the molding. Important factors in this respect are the location and type of the gate (Figs. 4-53 and 4-54).

The flow is largely governed by the shape and dimensions of the article and the location and size of the gate(s). A good flow will ensure uniform mold filling and prevent the formation of layers. Jetting of the plastic into the mold cavity may give rise to surface defects, flow lines, variations in structure, and air entrapment. This flow effect may occur if a fairly large cavity is filled through a narrow gate, especially if a plastic of low melt viscosity is used.

Jetting can be prevented by enlarging the gate or locating the gate in such a way that the flow is directed against a cavity wall.

The hot plastic melt entering the cavity solidifies immediately upon contact with the relatively cold cavity wall. The solid outer layer thus formed will remain in situ and forms a tube through which the melt flows on to fill the rest of the cavity (Fig. 4-55). This accounts for the fact that a rough cavity wall adds only marginally to flow resistance during mold filling. Practice has shown that only very rough cavity walls (i.e., sandblasted surfaces) add considerably to flow resistance.

For gate type and location, the points where two plastic flow faces meet must also be taken into consideration. If in these places flow comes to a standstill, which may be the case for flow around a core, premature cooling of the interfaces may cause weak weld lines. Although in practice sufficient strength may be obtained in such cases by good molding venting, high injection speed, and proper polymer and mold temperatures, the weld line can only be eliminated entirely by ring gating. Partial improvement is provided by a design in which the weld line has been shifted to a tab on the molding. This tab must be removed later, a step that involves additional cost, unless it is included in the design.

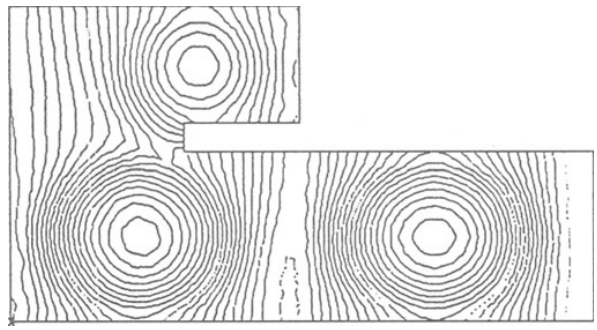


Fig. 4-54 Multiple-gate flow pattern.

DNA Genotek Inc. Exhibit 2029 Page 85
Spectrum Solutions LLC v. DNA Genotek Inc. IPR2023-01424

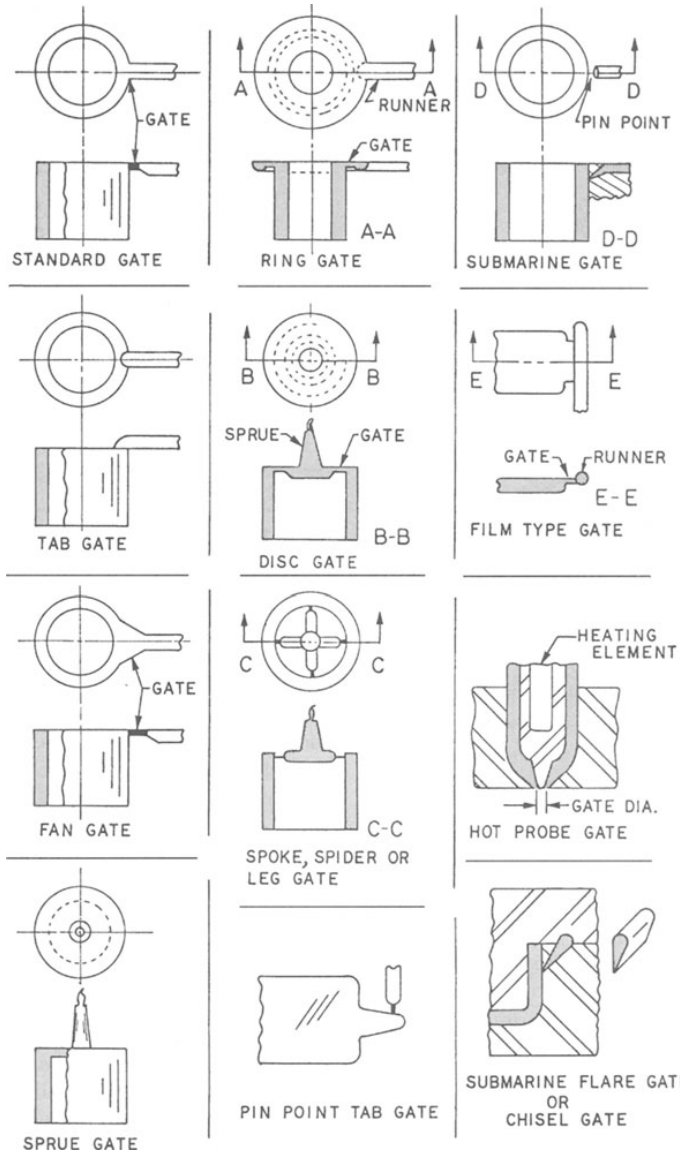


Fig. 4-55 Examples of different gate types.

Weld lines may also be formed at places where the plastic flow slows down, for example, at a place where wall thickness increases suddenly. In grid-shaped articles, weld lines are mostly inevitable. By correct gate location, the plastic flows may be arranged so as to meet on an intersection, in which case the plastic continues to flow, so that better strength is obtained than if the weld line were situated on a bar between two intersections.

The following gate types are usually employed, and each has its own advantage for application (Fig. 4-55):

1. *Direct gate.* For single-cavity molds where the sprue feeds material directly into

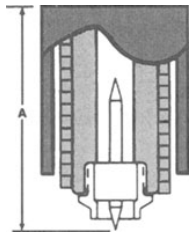


Fig. 4-56 Example of a pinpoint gate tip.

the cavity, a direct gate is applied. A standard bushing, bushing for an extended nozzle, or heated bushing may be used. Good rapid mold filling occurs.

2. *Pinpoint gate.* Generally used in three-plate and hot-runner mold construction, this provides rapid freeze-off and easy separation of the runner from the part (Fig. 4-56). The size of such gates may be as great as $\frac{1}{8}$ in., provided that the part will not be distorted during gate breaking and separation. A further advantage of pinpoint gating is that it can easily provide multiple gating to a cavity (for thin-walled parts), should such a move be desired for part symmetry or balancing the flow. It also lends itself to automatic press operation if the runner system and parts are arranged for easy dropoff. For a smooth and close breakoff, it is best to have the press opening at its highest speed at the moment when the plates causing the gate to snap are separating.

3. *Submarine (tunnel) gate.* Often used in multicavity molds, this type degates automatically, so it is particularly suitable for automatic operation. For multiple cavities, an angular gate entrance requires special care in machining during moldmaking, in order to ensure uniformity of the gate opening and consistency in the angular approach for a balanced runner system. The angle of approach is determined by the rigidity of material during ejection and the strength of the cavity at the parting line affected by the gate (Fig. 4-57). A flexible material will tolerate a greater angle of entrance than a rigid one. The rigid material may tend to shear off and leave the gate in place, thus defeating its intended purpose. On the other hand, the larger angle will give greater strength to the cavity,

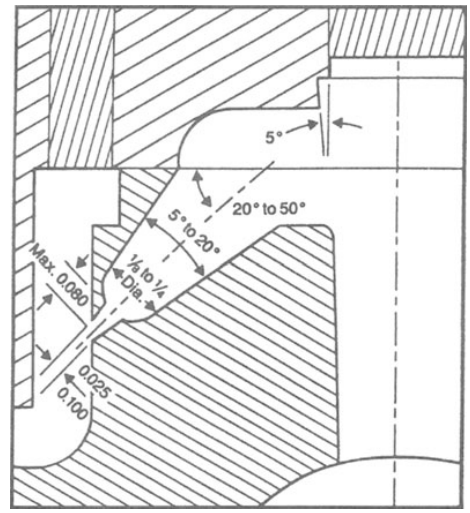


Fig. 4-57 Example of a tunnel gate.

whereas a smaller angle may yield a cleaner shearing surface.

4. *Tab gate.* This gate is used in cases where it is desirable to transfer the stress generated in the gate to an auxiliary tab, which is removed in a postmolding operation. Flat and thin parts require this type of gate.

5. *Edge gating.* Edge gating is carried out at the side or by overlapping the part. It is commonly employed for parts that are machine-attended by an operator. Normally, it is possible to remove the complete shot with one hand and in a rapid manner. The parts are separated from the runner system by hand with the aid of side cutters or, if an appearance requirement demands it, by such auxiliary means as sanders, millers, grinders, etc. When degating is performed with the aid of auxiliary equipment, it becomes necessary to construct holding devices.

6. *Fin or flash gate.* This gate is used when the danger of part warpage and dimensional change exists. It is especially suitable for flat parts of considerable area [over 3×3 in. (7.6× 7.6 cm)].

7. *Diaphragm-and-ring gate.* This gate is used mainly for cylindrical and round parts in which concentricity is an important dimensional requirement and a weld line is objectionable (Fig. 4-58).

DNA Genotek Inc. Exhibit 2029 Page 87
Spectrum Solutions LLC v. DNA Genotek Inc. IPR2023-01424

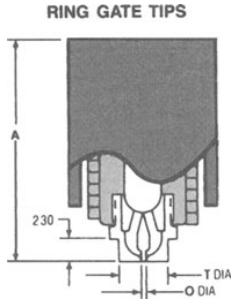


Fig. 4-58 Example of a ring gate.

8. *Internal ring gate.* This gate is suitable for tube-shaped articles in single-cavity molds.

9. *Four-point gate (cross gate).* This is also used for tube-shaped articles and offers easy degating. Disadvantages are possible weld lines and the fact that perfect roundness is unlikely.

10. *Hot-probe gate.* This may also be called an insulated runner gate and is used in runnerless molding. In this type of molding, the molten plastic material is delivered to the mold through heated runners, thus minimizing finishing and scrap costs.

Gates should always be made small at the start; they can easily be made larger but cannot so easily be reduced in size. Gate dimensions are important. Since the pressure drop in a system is proportional to the length of the channel, the land length of the gate should be as short as possible, but the strength of the metal may be a limiting factor, as may its machining method (with EDM, a razor edge can be used). On the average, 0.040 to 0.060 in. (0.10 to 0.15 cm) is a suitable length. The cross-sectional area for thin wall parts generally has a width and height of 50 to 100% of the runner cross section. (An example of a gate for thicker walls is shown in Fig. 4-59.) Equations are available for determining gate sizes of different shapes based on the plastic shear rate and volumetric flow rate.

When cavities are of different shot weights, the gate size of one cavity may be established arbitrarily as follows:

- For round gates:

$$d_2 = d_1\left(\frac{W_2}{W_1}\right)^{1/4}$$

- For rectangular gates (if we assume gate width is constant):

$$t_2 = t_1\left(\frac{W_2}{W_1}\right)^{1/3}$$

where d_1 = gate diameter of the first cavity (in. or cm)
d_2 = gate diameter of the second cavity (in. or cm)
t_1 = depth of gate in first cavity (in. or cm)
t_2 = depth of gate in second cavity (in. or cm)
W_1 = weight of first cavity component (oz or g)
W_2 = weight of second cavity component (oz or g)

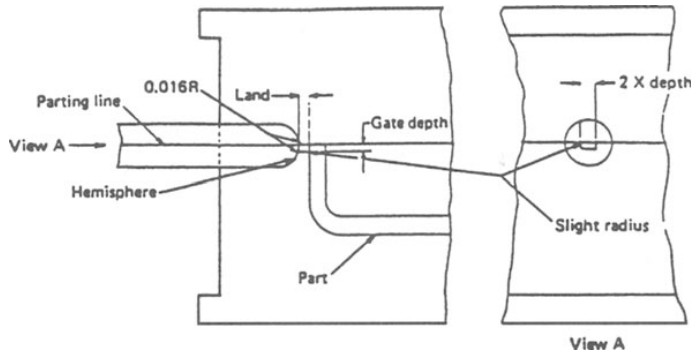


Fig. 4-59 Example of gate detail requirements.

Selecting hot-runner gates Hot runners offer a number of different gating styles, depending on plastic selection and the part design:

1. *Valve gating* uses a valve stem to produce mechanical shutoff at the gate, as opposed to pneumatic activation. With valve gating the gate size is normally larger and allows easier fill, creates less molded-in stress, allows for quick color changes, and is less likely to plug.

2. *Hot tip* is the most common style. It places a heated probe at the gate, supplying sufficient heat to keep the cold slug close to melt temperature and remelt it prior to injection.

3. *Thermal gates* deliver the plastic to the vicinity of the part and usually leave a cold sprue.

4. *Edge gating* allows gating on the side of a part, similar to a tunnel or submarine cold-runner gate. This type of gate shears itself off, leaving only a small mark.

Because the plastic structure characteristics of plastics vary considerably according to their crystallinity, thermoplastics are classified into the two main categories of crystalline and amorphous (Chap. 6). In the liquid phase, all are considered to be amorphous. Crystalline materials, during solidification, attain a degree of crystallization that is dependent on the processing parameters (time, pressure, and temperature) and that has a major effect on physical properties (100). Amorphous materials do not crystallize during solidification under any processing conditions. Figure 4-60 shows that in a crystalline material, the change between solid and liquid phases is sudden and easily discernible. In an amorphous polymer, the phase change is not so readily apparent, as the material remains in a softened state over a wide temperature range.

The temperature window available for processing crystalline thermoplastics is then much narrower than for amorphous materials. This can be calculated from Table 4-8, where the various molding parameters of amorphous and crystalline plastics are compared, including mold, average melting, and processing temperatures. The range below the processing temperature over which the plastic remains a liquid is determined by subtracting the average melting temperature from the hot-runner processing temperature. For example, let TD = (hot processing temperature-average melting temperature). Then for ABS we have TD = 250°C − 110°C = 140°C, and so on:

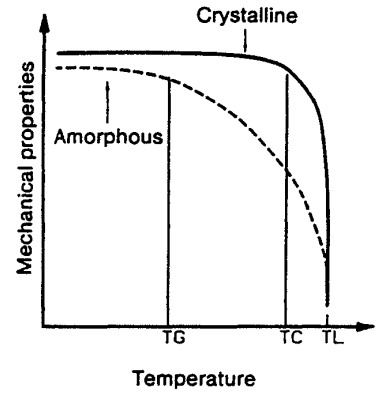


Fig. 4-60 Example of differences in the processing temperatures of crystalline and amorphous plastics.

Amorphous	Crystalline
ABS: TD = 140°C	PA 6: TD = 30°C
SAN: TD = 140°C	POM: TD = 10°C
PSU: TD = 115°C	PPS: TD = 40°C

This temperature difference is important in determining the style of gate, as it affects the rate of heat transfer required to optimize filling conditions under the shortest possible cycle time. The gate is a necessary evil. If it were possible, molding without gates would yield significantly better parts. The important action of the gate, as reviewed, is that it opens to let the plastic melt squeeze through and into the cavity. It closes once the cavity is properly filled. It must not only permit enough material to enter and fill the cavity, but also must remain open long enough to allow extra

Table 4-8 Thermoplastic injection temperatures

Thermoplastic (Abbreviation)	Average Melting Temp. (°C)	Material Structure[a]	Unreinforced		Glass-Fiber-Reinforced		
			Mold Temp. Temp. (°C)	Hot-runner Process Temp. (°C)	Fiber (wt %)	Mold Temp. Temp. (°C)	Hot-runner Process Temp. (°C)
PE	140	SEMI	25	250	30	40	230
PP	170	SEMI	35	255	30	40	245
PS	100	AMO	45	275	30	65	245
SAN	115	AMO	80	255	30	90	260
ABS	110	AMO	75	250	30	90	260
PMMA	100	AMO	70	245			
POM	181	CRYS	100	200	30	105	210
CA	227	AMO	75	235			
CAB	140	AMO	55	215			
CAP	190	AMO	65	225			
PETP	225	CRYS	140	280	30	140	285
PBTP	225	CRYS	35	255	30	90	243
PC	150	AMO	90	300	40	120	310
PA 6	220	CRYS	90	250	40	110	280
PA 6/6	255	CRYS	90	285	30	110	300
PA 6/10	215	CRYS	90	250			
PA 11	175	CRYS	60	230			
PA 12	175	CRYS	60	230			
PPO	120	SEMI	80	300	30	105	325
PVC	100	AMO	35	195			
PUR	160	AMO	35	205	30	50	215
PSU	200	AMO	150	315	30	160	385
PPS	290	CRYS	110	330	40	120	315
PES	230	AMO	150	350	40	150	360
FEP	275	CRYS	150	315	30	230	255
PAI	300	AMO	230	365			
PEI	215	AMO	100	370	30	150	420
PEEK	334	CRYS	160	370	30	180	380
LCP	330	CRYS	175	400	30	180	400

[a] CRYS = crystalline; AMO = amorphous; SEMI = semicyrstalline. For specific temperature values, use plastic material suppliers' information.

DNA Genotek Inc. Exhibit 2029 Page 90
Spectrum Solutions LLC v. DNA Genotek Inc. IPR2023-01424

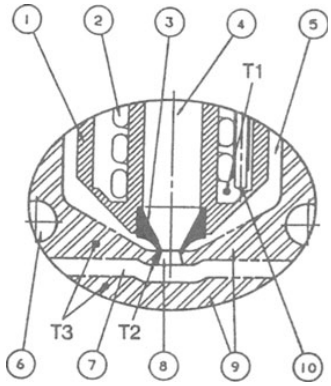


Fig. 4-61 Example of a hot-runner gate: 1, hot-runner nozzle; 2, heating element; 3, nozzle seal; 4, melt flow channel; 5, air gap insulation; 6, mold cooling; 7, mold cavity; 8, gate; 9, mold steel; 10, thermocouple (in copper pocket); T1, hot runner (processing) temperature; T2, gate-area temperature; T3, mold temperature.

plastic to accommodate shrinkage. (For example, nylon 6/6 has a volume contraction of about 8%.)

The opening and closing of the gate are, one way or another, thermally controlled. This includes mechanical shutoff gating, or valve gating, which is successful only because heat is transferred out of the pin, lowering the gate temperature. The thermal control of gate solidification is difficult and time-dependent. Figure 4-61 shows that the greatest upward pressure on the temperature occurs in the gate area identified as T2 in the nozzle. The nozzle is electrically heated and controlled, with its temperature set at the processing temperature. The mold cavity walls are set at a lower temperature (T3) and must not be affected by the heated nozzle, but thermally controlled by means of sufficient mold cooling.

In Fig. 4-62, consider the gate area to be in a state of thermal equilibrium, with no flow through the gate. In this example, the steady-state temperature of T2 is TS. It can be maintained at a specific level by providing a constant flow of heat from the nozzle to the mold cooling channel. It is the function of mold cooling to control the rate of heat transfer from not only the plastic, but also the hot-runner nozzle.

In the steady-state condition, the nozzle is the only heat source to the gate area that elevates TS above the mold temperature T3. This is represented by ΔTN in Fig. 4-62. The thermal gradient between two locations can be expressed by the following equation:

$$\Delta T = \frac{QL}{KA}$$

where Q = rate of heat flow
K = thermal conductivity
A = cross-sectional area
L = length of the heat-flow path

Under steady-state conditions, Q, L, and the gate diameter are constant. Therefore, the thermal gradient between the gate TS and nozzle T1 is a function of the following:

1. *Mold-to-nozzle contact area.* To maximize thermal separation, the contact area A must be minimized.

2. *Thermal conductivity of nozzle seals and nozzle tips.* For a large thermal gradient, the thermal conductivity K of the seal or tip must be low. The gate material should have a high K to give adequate heat flow from the material in the gate. This results in short cycle times.

As plastic begins to flow, rheological influences destroy thermal equilibrium. First, as the thermoplastic is forced through the gate, its velocity increases, causing a corresponding rise in both shear rate and kinetic energy; the smaller the gate, the greater these increases. Some of this kinetic energy is transformed into heat, which raises the local gate area temperature T2.

Second, T2 increases because of contact with the hot polymer melt flowing from the nozzle runner channel. Therefore, the temperature rise is a function of flow rate and velocity, as well as the diameter of the gate.

These two transient rheological influences create a rise in gate temperature T2 by an amount TA. The total increase in the gate temperature occurring during injection must not place T2 above the point at which thermal degradation could occur. Also, the temperature must not drop so far below the point at which the gate becomes plugged that normal

DNA Genotek Inc. Exhibit 2029 Page 91
Spectrum Solutions LLC v. DNA Genotek Inc. IPR2023-01424

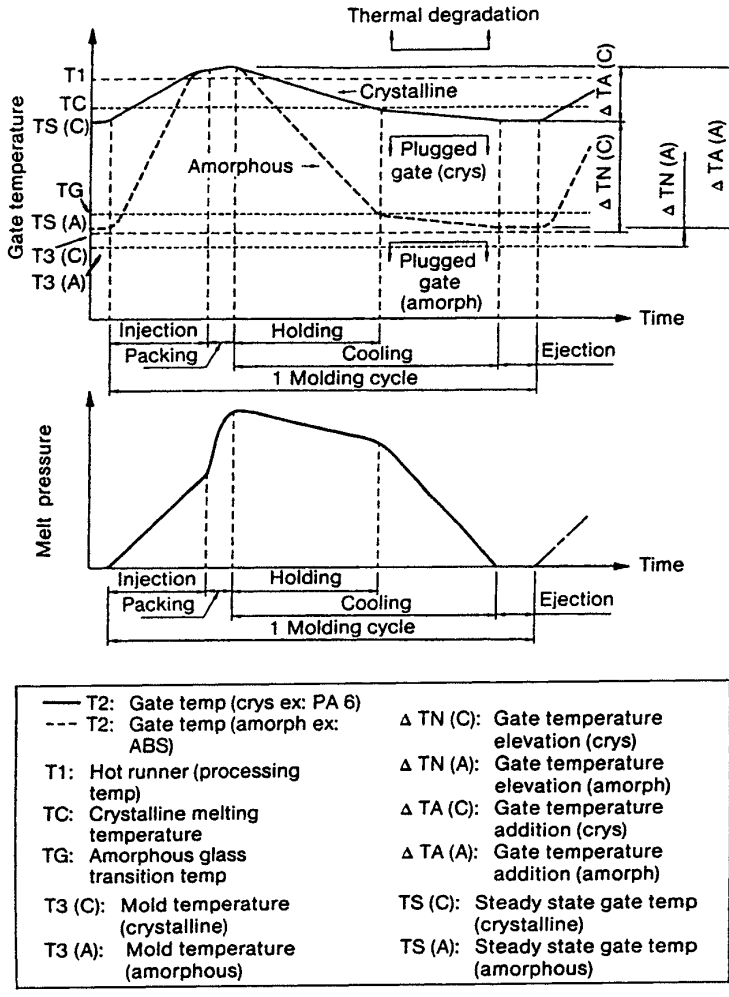


Fig. 4-62 Example of a process diagram showing processing conditions of crystalline and amorphous plastics in the gate area with temperature changes.

injection pressures cannot easily remove the plug with the next shot.

Selecting processing conditions for hot-runner gates A careful study of the gate-temperature-vs.-time graph (Fig. 4-62) makes it clear that different gating techniques are required to process amorphous and crystalline plastics. It shows that $\Delta TN(C) \gg \Delta TN(A)$. A steady transfer of heat takes place between the hot-runner nozzle and mold cooling (129). This action establishes an elevated steady-state gate temperature (TS = T3 + ΔTN). It is essential that the hot-runner nozzle end supply more heat to the gate area for crystalline than amorphous types, giving crystalline much higher steady-state gate temperatures, that is, $TS(C) \gg TS(A)$. Figure 4-62 also shows that $\Delta TA(C) \ll \Delta TA(A)$. Thus, additional heat is added to the gate via rheological influences, during injection, raising T2 by ΔTA. It is therefore essential that gate cooling for amorphous plastics be highly efficient, in order to dissipate the high heat generated during injection.

With poor cooling during this phase of molding, cycle times may be unacceptably

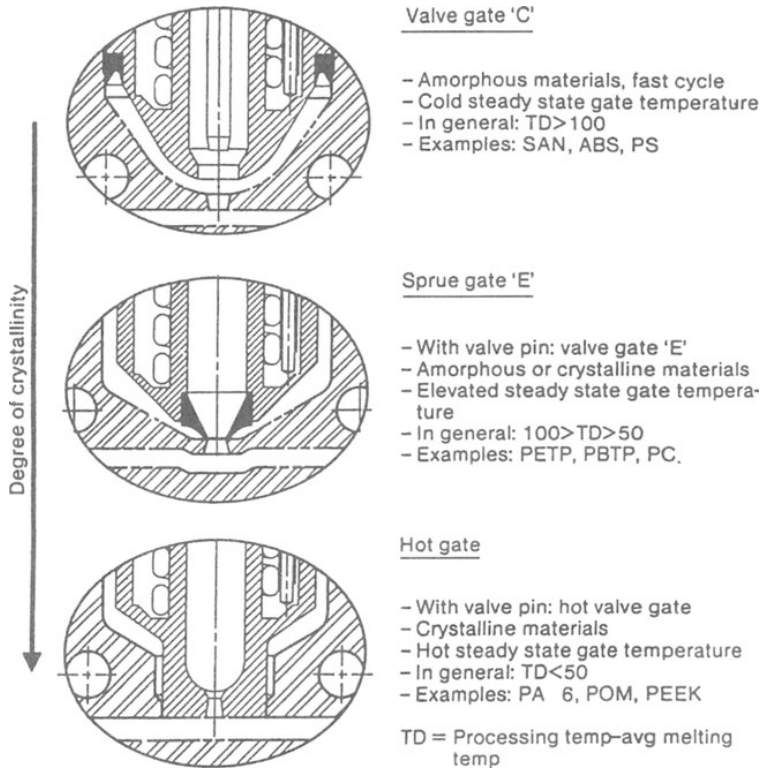


Fig. 4-63 Examples of heat transfer situations of the sprue gate.

lengthy. It is possible that no solidification will occur in the gate, resulting in stringing or drooling. However, if cooling in the gate area is too powerful for crystalline plastic, it is possible that the gate will freeze off prematurely, resulting in short shots and inadequate packing.

Gate size is also an important consideration. Small gates generate more heat, solidify more quickly, and are easier to degate. This is advantageous in the processing of amorphous plastic because of the low ΔTN and high ΔTA required during injection. Conversely, the required high ΔTN and low ΔTN necessitate a larger gate diameter for crystalline plastics. The example in Fig. 4-63 shows a hot gate specifically designed for the processing of crystalline plastics. Its rather massive nozzle end conducts heat away from the nozzle directly into the immediate gating area and provides the advantageous elevated temperature environment at the gate, that is, a large ΔTN.

Valve gate C (Fig. 4-63) was designed for the fast cycle processing of amorphous materials. Heat transfer from the nozzle tip is minimized by maintaining a plastic film around the nozzle tip, providing excellent thermal insulation between nozzle and gate steel. The absence of metal-to-metal contact results in the quick gate solidification required for dissipating a large ΔTA.

Many more gating methods, as explained by Mold Masters, Ltd., are available in the hot-runner industry. Another example is the sprue gate E (Fig. 4-63). These different versions provide suitable thermal behavior in the gate area to satisfy the wide range of processing requirements. In addition, the large quantity of gating methods allows the end user to select the style of gate mark that remains on the part. It is important to appreciate that

if the incorrect gate as well as other hot-runner components is used, processing problems usually exist that make it difficult to mold parts or extend the cycle time. Many of the past and present problems for mold designers of hot-runner systems have involved their inability to recognize that there are gates (etc.) which can only function certain ways.

Gate summary

Mold gate blush This is associated with melt fracture around the gate from stresses caused by process conditions or mold geometry. It is a blemish or disturbance in the gate area. To eliminate or reduce this problem, raise melt temperature, reduce injection speed, check gate for sharp edges, enlarge gate, and check that the runner system has a cold-slug well.

Mold gate, diaphragm A gate used in molding annular or turbular parts. The gate forms a solid web across the opening of the part. It is also called a disk gate.

Mold gate, direct A gate that has the same cross section as that of the runner.

Mold gate, fan An opening between the runner and mold that has the shape of a fan. This shape helps reduce stress concentrations in the gate area by spreading the opening over a wider area.

Mold gate, flash This is usually a long, shallow rectangular gate extending from a runner that runs parallel to an edge of a molded part along the flash or parting line of the mold.

Mold gate location The location of the gate must be given careful consideration, if the required properties and appearance of the molding are to be met. In addition, the location of the gate affects mold construction. The gate must be located in such a way that rapid and uniform mold filling is ensured. The gate must be so located that the air present in the mold cavity can escape during injection. If this requirement is not fulfilled, either short or burnt spots on the molding will be produced.

The gate should be located at the thickest part of the molding, preferably at a spot where the function and appearance of the molding are not impaired. However, the large-diameter gates require mechanical degating after ejection and always leave a mark on the product. With small or shallow moldings, the gate is sometimes located on the inside. However, this necessitates mold release from the direction of the stationary mold half, which interferes with effective cooling and generally increases mold cost.

Mold gate mark A surface discontinuity on a molded part caused by the gate through which material enters the cavity.

Mold gate, pinpoint A restricted orifice, 0.030 in. (0.76 mm) or less in diameter, through which melt flows. This small gate minimizes the size of the mark left on the molded part. The gate breaks clean when the part is ejected. Sometimes referred to as a restricted gate.

Mold gate, restricted See Pinpoint gate.

Mold gate, ring Used on cylindrical shapes, this gate encircles the core to permit the melt to move around the core symmetrically before filling the cavity, preventing weld line. There are external and internal ring gates in respect to the cavity.

Mold gate scar Most mold designs start out using a small gate(s). If the gate size is too large, scars in the gate area can occur. However, larger sizes permit faster fill and cycle time.

Mold gate size Gate size has a tremendous effect on the success or failure of attempts to produce high-quality parts economically. Plastic is a viscous liquid. The cooler the plastic, the more viscous it becomes. The more viscous it becomes, the more difficult it is to move it though very small gates. High injection pressure is then

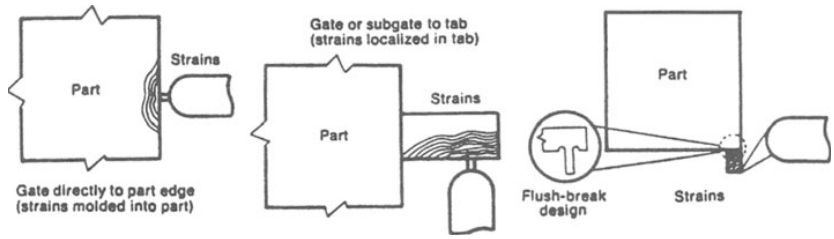


Fig. 4-64 Mold gate strains that can develop.

needed. The higher the injection pressure, the smaller the total area of the mold must be; otherwise, the pressure will result in flash (for TP and TS plastics).

Gate size is usually the critical factor that dictates the final mold-filling speed. Reducing melt viscosity by raising the melt temperature increases the mold filling rate, since there is less pressure drop across the gate. However, this can increase cycle time, since the heat put into the material must be removed in the mold. Although decreasing mold temperature helps achieve faster cycle times, it also requires additional injection pressure, which affects the clamp tonnage (depending on the projected filling area of a mold).

Mold gate, spider Refers to multigating of a part through a system of radial runners from the sprue.

Mold gate strain Figure 4-64 shows the effects of gating methods on molding strains.

Mold gate, submarine A type of edge gating where the opening from the runner into the mold is located below the parting line or mold surface. In the more conventual edge gating (as well as others), the opening is machined into the surface of the mold on the parting line. With submarine gates, the molded part is cut (by the mold) from the runner system on ejection from the mold. It is also called a tunnel gate.

Mold gate, tab A small removable tab of approximately the same thickness as the molded part, usually located perpendicular to the item. It is used as a site for edge gating location on parts with large flat sections. It also can be used as a site for gating, so that if any unacceptable blemishes appear, they will be on the tab, which is cut off (Fig. 4-64).

Mold gate types Figure 4-65 illustrates some gates with special descriptions; for additional gate illustrations, refer to Fig. 4-55.

Mold gate, valve VGs are used in injection molds and provide a wider processing window of operation and better product quality, eliminate gate freezing, and are cost-effective. Although it has been problematic, the VG is a matured device providing consistently reliable and productive processing of products ranging from commodity items to highly specialized components. A VG is a type of hot-runner gating system that uses a valve, usually a pin, to mechanically open and close the gate orifice. An actuating mechanism coordinates the movement of the pin with the molding cycle. To begin injection, the pin is retracted, opening the valve. After injection, the pin moves forward to close the valve for part cooling and ejection. The pin and its actuation mechanism are usually an integral part of the hot-runner nozzle. A wide variety of approaches to actuating the valve have been developed, including springs, adjustable air cushions, mechanical cams, pneumatic and hydraulic pistons, and designs that harness the injection pressure in the melt to actuate the valve(s).

In demanding molding applications that require packing plastic into molds to provide precise part weight and tolerances, the pin is actually driven into semisolidified gates. As long as the temperature is accurately controlled in the gate area, the gate is properly sized, and the closing is properly timed,

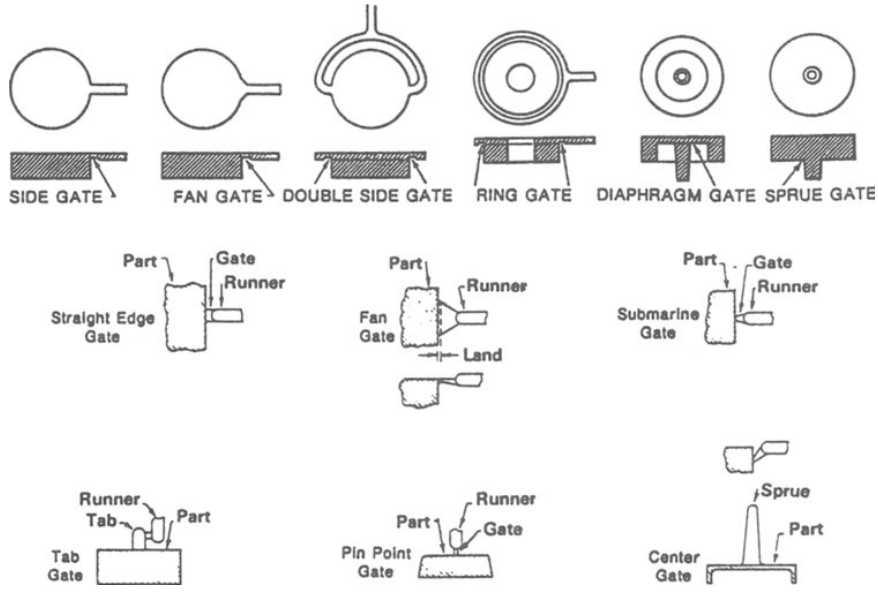


Fig. 4-65 Schematics of gates with cavities.

the valve will be closed by the action of the pin pushing through the soft core of plastic. This will close the gate precisely, without the risk of pin or gate damage. Regardless of the material used in any VG processing application, the gate must never be allowed to solidify (freeze) before the valve is mechanically closed. Otherwise, gate cosmetics will suffer and the gate itself may be damaged. The closing of the pin must always be accomplished above the melting point of a crystalline plastic, or well above the softening point of an amorphous plastic.

Correcting Mold Filling Imbalances in Geometrically Balanced Runner Systems

Flow imbalances in geometrically balanced runner systems have historically been attributed to variations in mold temperature and/or mold deflection. Through a series of molding trials and finite-element analysis, it has been proven that these imbalances result from nonsymmetrical shear distribution across the runner during injection. The resultant variations between cavities during molding include pressure, melt temperature, and mechanical properties of the molded parts. These effects can be significant, particularly when fine tolerances and tighter quality control are required. They further complicate the process settings, material, runner layout, and runner diameter. Hot-runner molds experience the same laminar flow and high-shear conditions as cold-runner molds. In addition the outer surfaces of the hot runner are heated by the runner manifold system, which can create additional variations across the flow channel. The following information (90) is a 1999 abstract on this subject by John P. Beaumont (Beaumont Runner Technologies, 5091 Station Rd., Erie, PA 16563, tel. 814-899-6390, www.meltflipper.com).

This review identifies an important means to expose the mold gremlin that has haunted the molding industry for decades. With the simple five-step process described, a mold-builder can clearly distinguish the source of variations found in a new or older mold. This can potentially eliminate the traditional time-consuming and costly process of repeatedly modifying gate, runner, and cavity sizes. The method described for diagnosing mold variations depends on the ability to recognize the multiple flows that exist in what was once

thought to be a naturally balanced runner system.

Shear-induced flow imbalances, developed in all multicavity molds utilizing the industry-standard naturally balanced runner systems, were not even identified or explained until late in 1997. These flow imbalances can be significant and now have been found to be the largest contributor to product variation between cavities. The understanding of this phenomena has not only led to the development of the Melt Flipper but has also provided a means for molders and moldbuilders to more clearly anticipate cavity to cavity variations and isolate their cause. The technique for isolating mold variations that is presented begins by isolating the cause of part variations into the two broad categories: external or internal to the mold.

External-to-mold influences on product variations can be expected to result primarily from variations in the plastic materials or the process. Such influences can be isolated by comparing parts produced within the same cavity over an extended run, or from run to run. If the external-to-mold factors (material and process) were identical, a part produced within a given cavity should be identical every time it is molded. The exception would be effects of mold wear on cavity or gate geometry that might occur over time. Variations in material can result from variations in the material as provided by the supplier or to the blending of regrind or other additives by the molder. Material variations can include factors such as molecular weight, molecular-weight distribution, and variations in additive percentages and distribution.

Potential process variations that can occur are almost too numerous to mention. Some of the more obvious include material drying, melt temperature, injection rates, pack pressure, pack time, mold coolant temperature, and flow rate. Additional variations between shots can be tied to atmospheric conditions (temperature and humidity) and the human inconsistencies introduced by the operator. The sensitivity of part size, weight, and mechanical properties is effected by so many variables that it is unreasonable to expect exact duplication of a part from shot to shot.

Internal-to-mold variations is generally those attributable to the moldbuilder. These can be found by comparing the parts produced from different cavities within a single shot. Differences in parts produced within a single shot are clearly distinguishable from the shot-to-shot variations created by the external influences. Averaging the variations occurring between a given cavity over two or three shots virtually eliminates the potential variations due to temporary clogging of a gate by an unmelted pellet, to contaminants, etc.

The variations created within a given shot can be further broken down into three subcategories. Despite the geometrical balance, in what have traditionally been referred to as naturally balanced runner systems, it has been found that these runners can introduce a significant variation into the melt conditions delivered to the various cavities within a multicavity mold. These variations can include the subcategories of melt temperature, pressure, and material properties. What must be recognized is that conventional geometrically balanced runners actually create multiple flows much like the old tree-branching-type runner. These in turn produce multiple families of parts in the mold. There are normally two flows in an 8-cavity mold, four in a 16-cavity, eight in a 32-cavity, etc.

It is important to be able to identify the different flows that exist in a geometrically balanced runner. The flow fed by the outer laminates of the primary runner is typically the dominating flow. Parts produced from this flow are typically larger and heavier. In a mold with two flows, the outer branching flow is fed by the center laminates of the primary runner. If there are more than two flows, as in a 16- or 32-cavity mold, only the dominating flow is obvious. The remaining flows are all fed from inner laminates of the primary runner, and it becomes less obvious which will progressively become subordinate flows. The numbering of these flows is therefore more arbitrary. In a mold with parting-line injection, a typical 4-cavity mold will have two flows, an 8-cavity mold will have four flows, a 16-cavity will have eight flows, etc.

Once the flow-induced variations have been identified, one can isolate the variations

produced by the physical makeup of the mold. These are variations that would occur within a given shot, and they can be compared. As parts within a given flow and given shot should be identical, any measurable differences between parts can only result from variations in the physical makeup of the mold and the cooling of the mold. These part variations can be caused by the runner layout; differences in the size of cavities and gates, in runner lengths, and in runner diameters; venting, etc.

Variations between cavities within a given shot can also be caused by variations in the cooling between the different cavities. This variation would result from the circuit networking or water flow rate. The network could cause different amounts of water to be delivered to each cavity or the accumulation of heat in the water as it flows through the circuit. The largest effects of cooling differences between cavities occur during packing and cooling phases of the molding cycle. These effects might include surface finish, shrink, and warp. This conclusion comes from studies that show that mold temperature has a minimal effect on mold-filling imbalances. Therefore variations in mold temperature would have a minimum impact on the weights of samples molded from partially filled cavities (no packing stage). These partially molded parts are formed with only a filling phase. The shear-induced flow imbalance and dimensional variations in the mold steel are therefore the only possible causes of any variation in weights.

The best method for isolating variations introduced within the mold is to compare the weight of short-shot-molded parts from each cavity. An additional benefit of the short-shot method is that it helps separate out any cooling variations between cavities. If there is an imbalance created by any variations in the mold, it will be clearly evident. For example, an imbalance that causes a cavity to fill 20% sooner than another cavity will be evident by comparing the weight of short-shot-molded parts from each cavity. The leading part should be approximately 20% heavier. If on the other hand you allow the cavities to fill completely and fully pack out, the difference between parts will be masked by the smaller difference in cavity weights and thereby more difficult to isolate. In the fully packed-out cavity, the leading flow will fill the first cavity and the remaining flows will eventually fill their cavities one by one. The parts will then be packed out under a high pressure. When the parts are then weighed and compared, their difference will be minimized and may be less than 0.2%.

Hot-runner molds complicate the task of isolating molding problems, as variations between parts, both shot to shot and within a given shot, can be introduced by temperature variations in the manifold and hot drops. Temperature variations between the drops and along the manifolds would result in variations between cavities during a single shot. This has been characterized earlier as an internal-to-mold variation. However, the temperature within these same regions (drops and manifold) can drift with time, which will cause shot-to-shot variations. This has been characterized earlier as an external-to-mold variation. Therefore the hot manifold introduces both internal-to-mold and external-to-mold variation. This combined effect makes it more difficult to isolate the variations created by steel dimensions and shear-induced flow imbalances.

Isolating Mold Variations in Multicavity Molds

Studies were performed on over twenty molds to evaluate the best technique for isolating cavity-to-cavity variations in multicavity molds. These studies were based on data collected from current production molds and several test molds from Pennsylvania State University's plastics processing lab in Erie. The simple five-step process was developed from these studies, for which much of the detailed procedure and data have been documented. The following procedure assumes a geometrically balanced runner design.

Step 1 concerns mold samples. For a given mold, the plastic material should be conditioned per supplier specification and the process established per normal procedure. If there is no history of running the mold, consider finding the fill rate by generating a curve

of relative viscosity vs. relative shear rate, using your molding machine, as described by John Bozzeli (117). This method identifies the injection molding velocity from the lowest pressure to fill. Having established a reasonable process for this mold, reduce the screw feed and set the hold pressure and hold time to the minimum value that the process controller permits (zero where possible). Screw feed should be reduced until the best-filling cavity in the mold is about 80% full. That cavity will reduce the potential of hesitation effects or venting issues from masking the imbalance. The original injection rate should remain constant.

Step 2 involves collecting all the molded parts from a single shot and weighing them individually. This can be done immediately, as the samples do not need to be conditioned.

Step 3 involves identifying the parts molded from flow 1 (4 parts in molds with eight or more cavities. 2 parts in a four-cavity mold). Contrast the weights of these parts with each other to determine the variation resulting from dimensional differences in the mold steel.

Step 4 involves identifying each of the other flows and repeating step 3. This will isolate the effect of the dimensional variations in the mold steel on each of these flow groups.

Step 5 involves identifying the parts molded from flow 1 and determining their average weight. Contrast this with the average weight of the four parts molded from flow 2. The difference is due to the shear-induced variation created within the runner. This variation is independent of dimensional difference in the mold steel.

Detailed studies on several molds indicate that it is best to contrast weights of parts when the best-filling cavities (flow 1) are between 80 to 90% full. The actual percentage is dependent on the part geometry, gating, and venting. However, for simplicity, it is suggested to contrast the part weight between the various cavities when the best-filling cavities are 80% full. There will be some cases where this may be difficult due to the requirement of ejecting the molded part.

Mold Components

The following information is a guide regarding some of the many components in molds (Figs. 4-10, 4-11, and 4-66). Also the

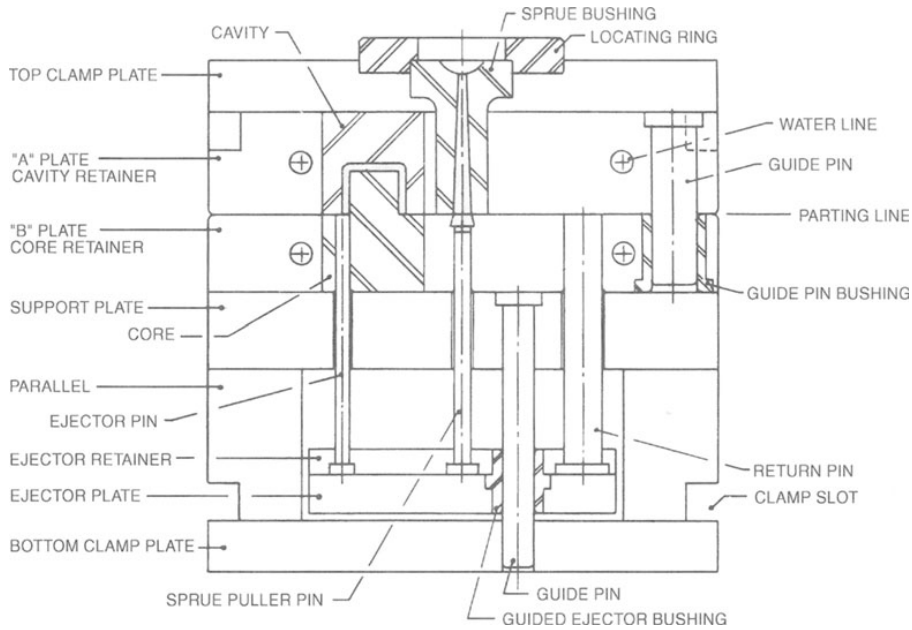


Fig. 4-66 Mold nomenclature.

DNA Genotek Inc. Exhibit 2029 Page 99
Spectrum Solutions LLC v. DNA Genotek Inc. IPR2023-01424

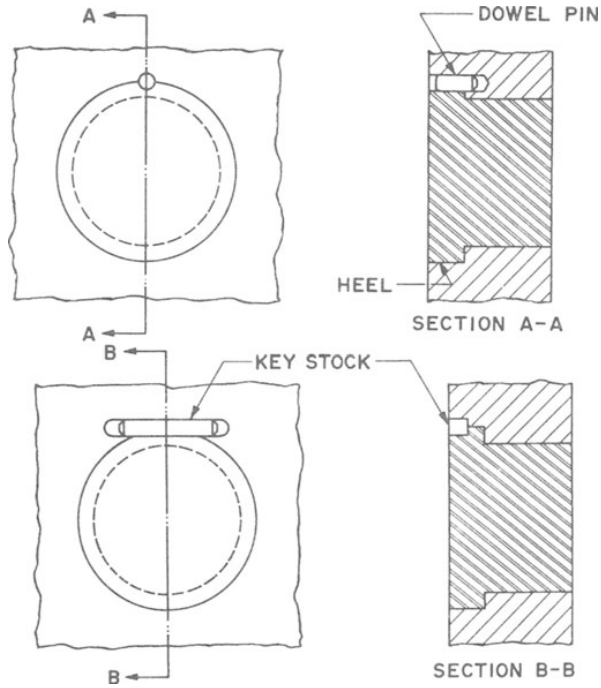


Fig. 4-67 Example of a key stock locking device.

reader is referred to the section on Preengineered Molds at end of this chapter, which also addresses components. In the large single-cavity molds, the entire cavity and core plates usually form the mold cavity. In smaller and multiple-cavity molds, core and cavity inserts are mounted on or in the various plates of the mold base. When various components are mounted on a plate, the plate may be called a yoke or chase. A simple method is to mount a cavity directly to the clamping plate with screws and dowels. Generally, two dowel pins are used, spaced far enough apart to prevent any twisting of the mating mold cavities. Two or more cap screws hold the cavity spacing firmly to the clamping plates.

More often, cavity blocks are retained in pockets machined in the mold plates. There are types such as the window pocket, window pocket with counterbore, blind pocket, channel shape, and circular pockets. Cavity blocks that are in square or rectangular pockets will not turn during the molding process. Blocks mounted in circular pockets must be locked to keep them from turning. Sprue bushings are locked to keep the runners in the sprue bushing and runner plate aligned. Ejector pins that eject at an irregular surface of the part must also be keyed. Figure 4-67(a) illustrates a method of locking sprues, circular blocks, pins, etc., using a dowel pin as a key. Figure 4-67(b) shows a square piece of key stock used as the locking device. This type is not used to lock sprue bushings. Figure 4-68 illustrates the use of the pointed set screw as a lock.

Ejector Systems

The conventional mold ejector system moves between the clamp plate and support plate in a space provided (Figs. 4-69 and 4-70). The ejector plate and pin plate are guided by return pins that ride on bearing surfaces in the support plate and core plate. The ejector plate carrying the ejector pins must move freely in the mold. In order to reduce undue friction, clearance is provided in the support plate, core, or cavity blocks to within $2\frac{1}{2}$ to 3 times the pin diameter of the parting surface.

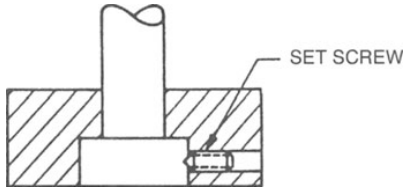


Fig. 4-68 Example of set-screw locking device.

The number and location of the ejector pins are determined by the size and shape of the piece part. Most mold bases use four or more return pins, one of which is offset so the ejector unit can be assembled into the mold in only one position. The clearance holes in the support plate, core plate, and core block are drilled larger than the diameter of the ejector pins. The same clearance is provided for the sprue puller pin. The holes in the ejector retainer plate are drilled larger than the diameter of the ejector pins. The ejector retainer plate is counterbored larger than the diameter of the head of the ejector pin. This

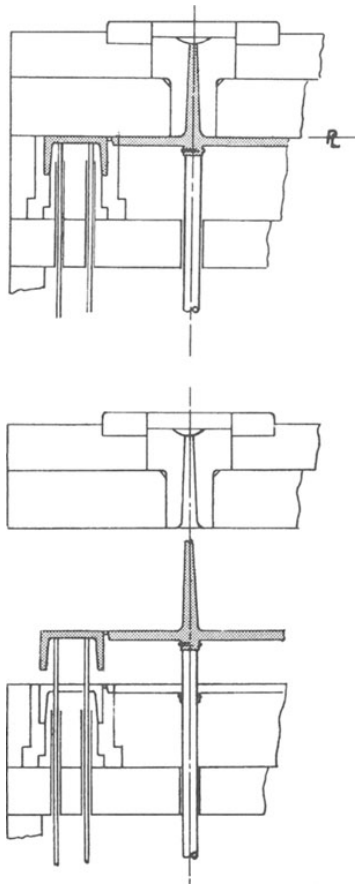


Fig. 4-69 Example of ejector system used in a two-plate mold.

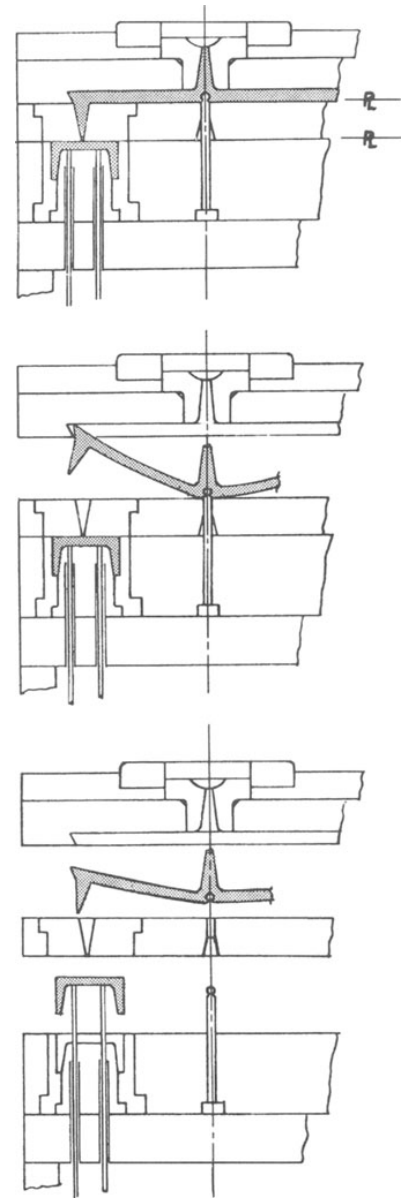


Fig. 4-70 Example of ejector system used in a three-plate mold.

DNA Genotek Inc. Exhibit 2029 Page 101
Spectrum Solutions LLC v. DNA Genotek Inc. IPR2023-01424

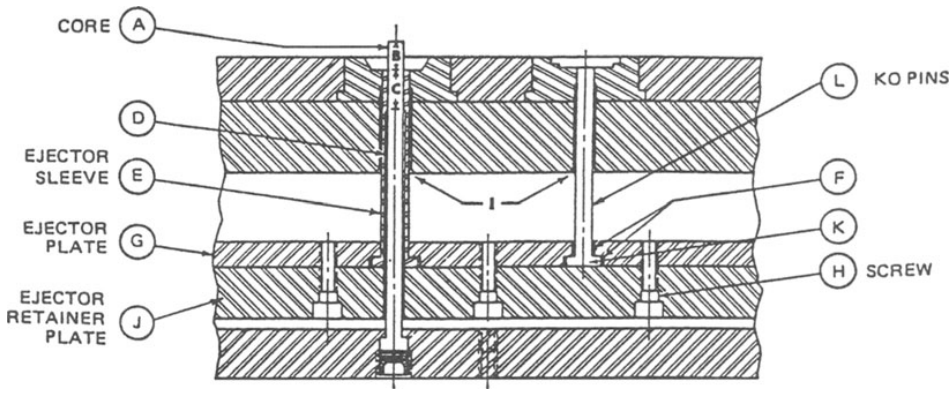


Fig. 4-71 View of ejector (KO) pins, ejector plates, and stripper pin and sleeve.

counterbore is deeper than the height of the pin head. Minimum clearance should be provided between the rail and side of the ejector unit. If pillars are used, the holes drilled through the ejector unit should allow 0.06 in. (0.15 cm) of clearance per side. Pillar supports are machined higher than the height of the parallels.

Sticking in a mold often is related to the elasticity of steel. When the injection pressure is applied to the molten plastic, the steel of the mold deforms. When the pressure is relieved, the steel will return to its original position and then act as a clamp on the plastic. Additionally, packing causes sticking by increasing even more the adhesive forces between the plastic and mold. Very often, a reduction of the injection pressure and/or the injection forward time will eliminate the problem. Packing is also common in multicavity molds where the individual cavities do not fill equally. One cavity will seal off first, and the material intended for that cavity will be forced into other cavities, causing overfilling.

Ejector mechanisms Ejector pins are made either from H-11 or a nitriding steel. They have a surface hardness of 70 to 80 Rc, to a depth of 0.004 to 0.007 in. (0.010 to 0.018 cm). The inside core is tough. The heads are forged and annealed for maximum strength, and they are honed to a fine finish. They come in fractional and letter-size diameters, each being available in a 0.005-in. (0.013-cm) oversized pin. They are used when the knockout holes in the cavity or core are worn and flash occurs around the pins. The right side of Fig. 4-71 shows the way a knockout pin (L) is mounted. The ejector plate is drilled and countersunk. The pins are held in by screwing the ejector retainer plate (J) to the ejector plate (G). The ejector pins, ejector sleeves, sprue puller, and return pins are all located in this plate. This construction facilitates assembly of the mold because the pins can be entered one by one into the cavity plate. It is often difficult to assemble large molds with a great number of ejector pins if the construction does not allow the pins to be inserted individually. This construction also makes it possible to remove one or two knockout pins without removing all of them.

There is nearly always a slight misalignment between the holes and cavity plate and the ejector plate. Therefore, it is important to leave a clearance of from $\frac{1}{64}$ to $\frac{1}{32}$ in. (0.04 to 0.08 cm) around the heads of the pins and at least 0.002-in. (0.005-cm) clearance at K. This will permit the pins on the counterbore to find their proper location when the mold is assembled. Return pins should be used only to return the knockout plates, not to guide them. The knockout plates should be on their own leader pins and bushings. Chamfers at point I are helpful for easy insertion of the pins. The holes for the ejector pins should be relieved to within a fraction of an inch of the face of the cavity or core to facilitate the alignment and

DNA Genotek Inc. Exhibit 2029 Page 102
Spectrum Solutions LLC v. DNA Genotek Inc. IPR2023-01424

operation of the pins. The top of the knockout pins will leave a circle on the molded parts.

Ejector sleeves Ejector sleeves, labeled E in Fig. 4-71, are preferred when molded parts have to be stripped off round cores. They are subjected to severe stress and wear, so their inside and outside surfaces must be hard and finely polished. If they are not sufficiently hard and are of different hardness, scoring of both the cavity and core may take place. Additionally, both the cavity and core must be of a different hardness from the ejector sleeve. Two parts of equal hardness, regardless of how hard they are, will scour. The lower portion of the sleeve should be drawn to obtain maximum toughness, whereas the upper part should be left hard for the full length of the ejector movement.

The outside diameter of the sleeve should be about 0.001 to 0.002 in. (0.0025 to 0.0050 cm) smaller than the hole in the cavity. An equal clearance between the core and sleeve should be maintained for a distance *C*. The inside diameter of the sleeve should be about $\frac{1}{64}$ to $\frac{1}{32}$ in. larger than the core, leaving a clearance as indicated by *D*. The core (A) should be dimensioned so that the portion that extends into the molded article (distance *B*) is at least $\frac{1}{64}$ in. smaller in diameter than the lower part. If this is not done, the reciprocating movement of the sleeve will damage the fine finish of the core. Distance *C* should be at least $\frac{3}{8}$ in. (0.95 cm) longer than the entire movement of the ejector plate. If the clearance extends too far, the shoulder and end of the core may be damaged when the sleeve is retracted. It is also important to leave a clearance of $\frac{1}{64}$ in. (0.040 cm) around the outside of the sleeve. This clearance, however, should not extend too far, because it is necessary to have a bearing at least $\frac{1}{2}$ in. (1.27 cm) long at the cavity.

Early ejector units A typical mold base accessory from DME that provides an early ejector return unit is shown in Fig. 4-72. Whenever a mechanically operated cam slide passes over an ejector pin, the ejector plate must be returned early (before the mold is closed); otherwise, the returning cam will slide and hit the ejector pins with mold damage. This is not the case if the machine or mold has air or hydraulically operated knockout plates. To prevent this problem in other types of molds, an early ejector return unit can be used. The unit consists of a bushing, post with slidable cam fingers, and cam actuating pin. The bushing is installed in the B plate, the post is attached to the ejector plate, and the cam actuating pin is installed in the A plate.

In operation, the early return of the ejector plate is accomplished while the press is closing, by the cam actuating pin pushing against the projecting cams on the post (and thus returning the ejector plate) until the cams are released into a matching countersink, which happens when the ejector plate is fully back. The cam pin then passes on through as the mold continues to close. Timing is regulated by adjusting the length of the pin.

Ejector Pin Strength

Breakage of small-diameter ejector pins is common, especially when certain materials are molded into parts with confined configurations. Besides producing defective molded parts, broken pin ends can sometimes damage cavity walls or elongate ejector pin holes. Ejector-pin failures not repaired immediately may cause progressive overloading and failure of remaining pins. In addition to the modest costs of replacement pins, the total costs of broken ejector pins include molding machine downtime, extra setup time, and toolroom repair time (7,589).

Ejector-pin breakage is categorized here either as within the ejector housing area, or as parting-line-area breakage—that is, inward from molded-part contact. Ejector-housing-area failures are usually buckling failures that take place upon ejection initiation, whereas line failures may be due to either buckling or deflection. Close examination of these kinds of ejector-pin failures usually reveals conditions to which slender-column formulas can be applied.

Slender-column formulas, developed by Leonard Euler (Swiss mathematician, 1707–

DNA Genotek Inc. Exhibit 2029 Page 103
Spectrum Solutions LLC v. DNA Genotek Inc. IPR2023-01424

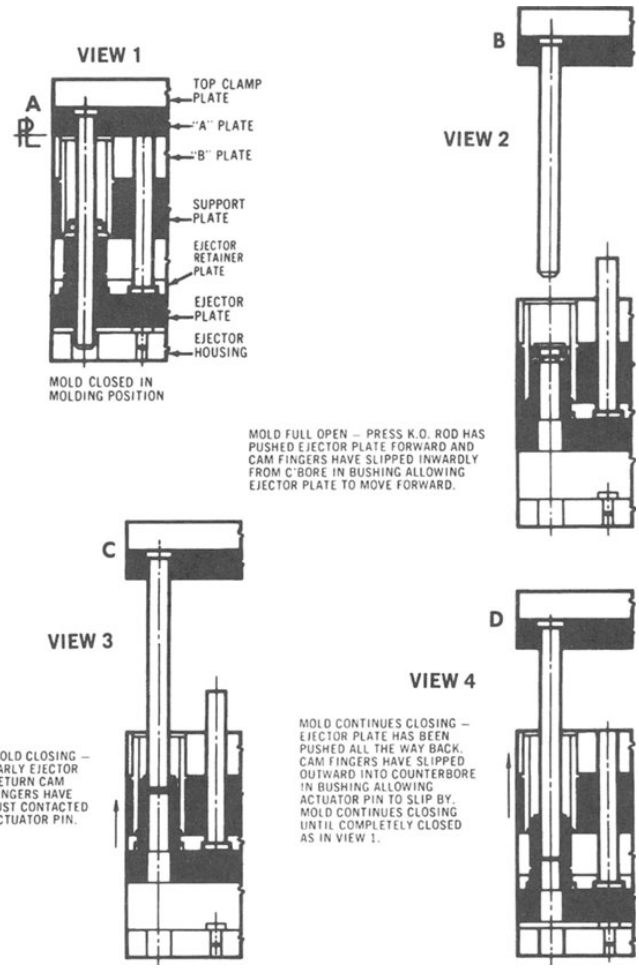


Fig. 4-72 Examples of a positive early ejector system and operating sequence for lighter ejector systems.

1783) and found in most machine and tooling handbooks as well as strength-of-materials textbooks, have seldom been applied to mold design. Ejector-pin analysis during mold design can, in many cases, prevent expensive repairs and rework. Likewise, formula analysis of ejector-pin conditions in existing high-breakage molds may reveal simpler, more effective, and less costly remedies and improvements.

High ejection forces on ejector pins may be due to, but not limited to, the following:

- The use of stiff materials that have a particularly strong tendency to cling to mold walls, such as polycarbonate
- Tendencies of soft elastomeric materials to compress and expand due to ejector pin forces, especially in molded rib sections
- Insufficient draft
- Configuration constraints such as undercuts, deep ribs and posts, and extensive coring
- Such molding conditions as overpacking due to increased pressures or temperatures
- Insufficient size of ejector pins
- Insufficient number of ejector pins

Euler first published his critical-load formula for columns in 1759. It is usually expressed in the following form:

$$F = \frac{m\pi^2 EI}{I^2} = \frac{m\pi^2 EA}{(I/k)^2}$$

where F = collapsing load on the column in pounds
I = length of the column in inches
A = area of the section in square inches
k = least radius of gyration $= I/A$
E = modulus of elasticity $\cong 30$ million psi (20.7 GPa)
I = least moment of inertia of the section
m = a constant depending on the end conditions of the column (see Fig. 4-73)

Euler's formula is strictly applicable to long and slender columns, for which the buckling action predominates over the direct compression action, and thus makes no allowance for compressive stress. The slenderness ratio is defined as the ratio of the length I to the radius of gyration k.

When the slenderness ratio exceeds a value of 100 for a midsteel column, failure by buckling can be expected. Columns of stiffer and more brittle materials will buckle at lower slenderness ratios.

The constant factor m in Euler's critical-load formula clearly shows that the failure of a column depends on the configuration of the column ends. Figure 4-73 shows simplified schematic column types and their theoretical curves of flexure. These four types are:

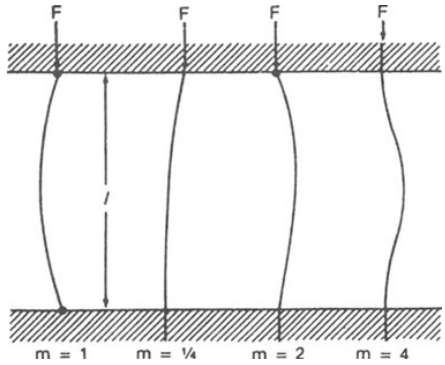


Fig. 4-73 Types of end connections for columns.

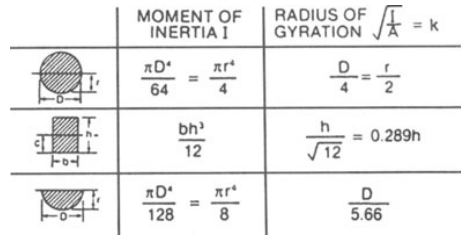


Fig. 4-74 Moments of inertia and radii of gyration.

- Both ends pivoted or hinged ($m = 1$)
- One end fixed and the other free ($m = \frac{1}{4}$)
- One end fixed and the other pivoted ($m = 2$)
- Both ends fixed ($m = 4$)

Figure 4-74 shows cross sections of the three most common ejector-pin configurations, which include the round ejector pin, modified half-round pin, and ejector blade. Formulas for each respective moment of inertia I and radius of gyration k are given for convenience.

With the above formulas, the buckling force F can be calculated for an ejector-pin configuration. This force should exceed a calculated load to punch through or shear the molded part; a safety factor of 4 is recommended.

Ejector-housing applications Figure 4-75 illustrates typical housing applications of the two most common column end types: one

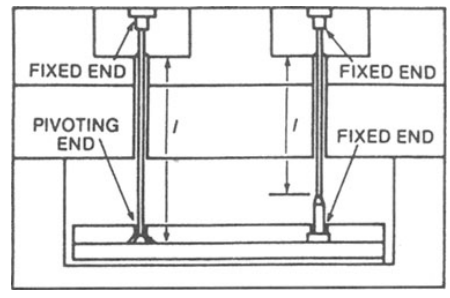


Fig. 4-75 Determination of column length with one end fixed and one end pivoted (left) and with both ends fixed (right).

Table 4-9 Slenderness ratio (l/k) of round ejector pins

Column Length (in.)	l/k at Diameter (in.)						
	0.031	0.047	0.0625	0.078	0.083	0.125	0.1875
1.0	128	85	64	51	43	32	21
1.5	192	128	96	77	64	48	32
1.75	224	149	112	90	75	56	37
2.0	256	171	128	102	85	64	43
2.25	288	192	144	115	96	72	48
2.5	320	213	160	128	107	80	53
3.0	384	256	192	154	128	96	64
3.25	416	277	206	166	139	104	69

end fixed and the other end pivoted (loose bevel head), and both ends fixed. A fixed end may allow axial movement, although the pin shank is surrounded. The illustrations in Fig. 4-75 show the interpretations in establishment of lengths for calculation purposes.

Table 4-9 lists values of slimness ratios (l/k) for small-nominal-diameter ejector-pin applications of typical column lengths. Table 4-10 shows ejector-blade values for comparison. [The slimness ratio for a blade depends on column length and thickness, but not width A (see Fig. 4-74)].

Figure 4-76 A shows a typical mold problem that usually occurs due to the difficulty of drilling small-diameter holes through thick plates and mold inserts. Figure 4-76B through D illustrate possible solutions.

Most failures within the ejector-plate section occur because the slenderness ratio of susceptible ejector pins exceeds 100 in the mold-open, ejector-plate-back configuration.

Table 4-10 Slenderness ratio (l/k) of ejector blades

Column Length (in.)	l/k at Thickness (in.)			
	0.030	0.050	0.080	0.125
1.0	115	69	43	28
1.5	173	104	65	42
1.75	202	121	76	48
2.0	231	139	87	55
2.25	260	156	97	62
2.5	288	173	108	69
3.0	346	208	130	83
3.25	375	225	141	90

The prudent mold designer devises ways to reduce or limit the slenderness ratio.

Parting-line applications Ejector-pin damage at the mold parting line is likely due to the following conditions, which are usually characterized by an m value of only 0.25 (see Fig. 4-73):

- The slenderness ratio exceeding a value of 100 before stiff ejection forces cease, as in a long core with reverse draft
- The mold closing on part or surface flash (with critical force) when the ejector plate is fully forward and the slenderness value above 100

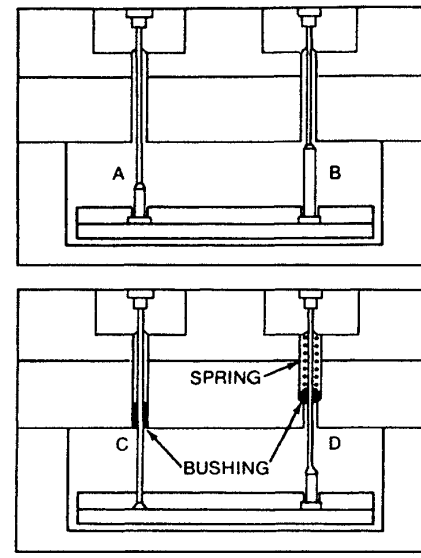


Fig. 4-76 Example problem and possible solutions.

- Ejector-plate deflection, possibly due to numerous ejector pins, insufficient lubrication, tight fits, and/or weakness due to support post holes or other causes
- Insufficient return-pin size, number, and locations; most applicable to modular quick-change mold-insert units and home-made mold bases
- Short shots or breaking of the molded part, causing uneven loading of ejector pins and possible tilting/binding of the remaining part

Design solutions to these problems often require using the most and largest-diameter ejector pins possible. Some other solutions include limiting the slenderness ratio to a value of less than 100 by restricting the ejection stroke, or increasing the draft or stepping the wall section.

Sprue Pullers

Common designs of sprue pullers can be used. Avoid any that restrict the flow of the plastic. A 5°-reverse-taper sprue puller, as shown in Fig. 4-77, works well.

Side Actions

Holes and undercuts produced in the molded piece part at an angle other than parallel to the press movement make it impossible to eject by the conventional ejection system. Hydraulic cylinders that are mounted to the mold base pull out these core pins before the mold opens at the parting line (Figs. 4-78 to 4-81). Keys or T slots keep the

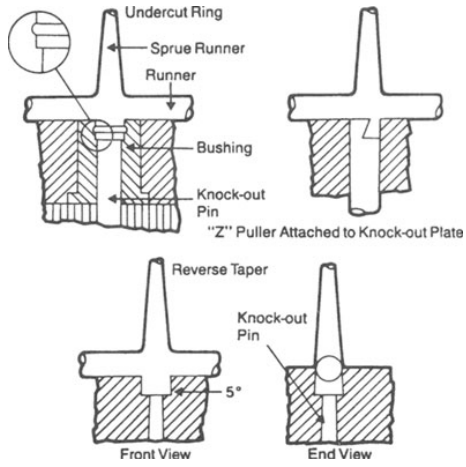


Fig. 4-77 Examples of sprue pullers.

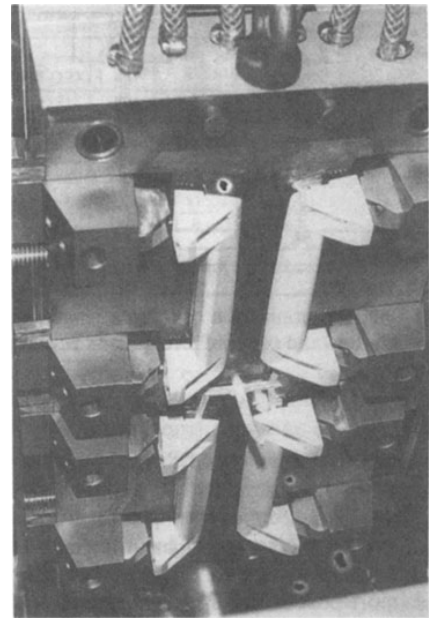


Fig. 4-78 Side-action mold fabricating car door handles.

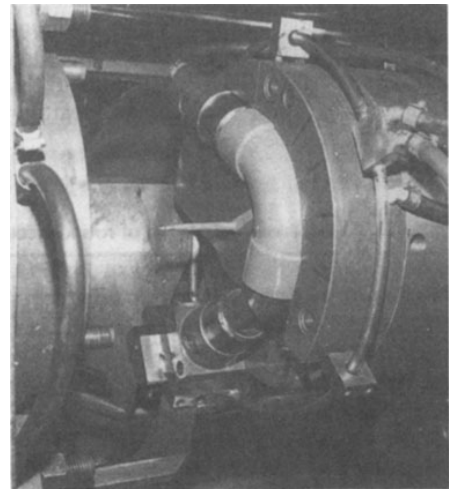


Fig. 4-79 Side-action mold fabricating pipe fitting.

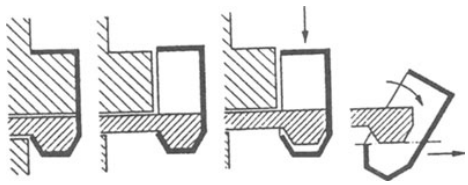


Fig. 4-80 Side action in a mold that is a "slide within a slide."

core pins in position. Extremely long, unsupported, small core pins can be piloted in the plunger, whereas larger core pins are butted against the plunger.

Angle Pins

Angle pins are also used to pull out core pins. Angle pins use the normal movement of the molding machine to remove the core pin as the mold opens at the parting line. Motorized racks and pinions are used to unscrew threaded cores. Figure 4-82 shows the use of an angle pin with the mold in the closed position. The lock that is at an angle of 5° greater than the angle of the angle pin butts against the back surface of the slide. This positive locking device is used to ensure that the core pin is in the proper position and keeps the pin from retracting as the material is injected into the mold. As the mold opens, the piece part stays with the movable portion of the mold.

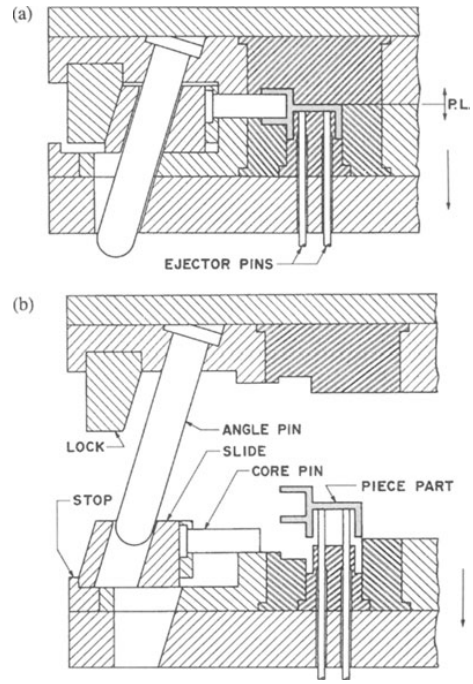


Fig. 4-82 Example of an angle pin with (a) mold closed and (b) mold open.

The angle pin forces the slide to move toward the outer edge of the mold, pulling the core pin from the piece part. Once the core pin is free of the piece part, ejector pins push the piece part from the plunger in the conventional manner (Fig. 4-82). The slide is kept

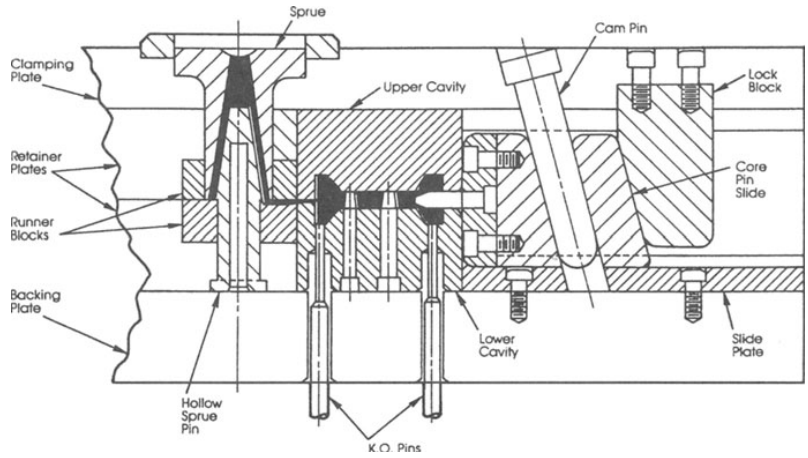


Fig. 4-81 Mold includes side core action.

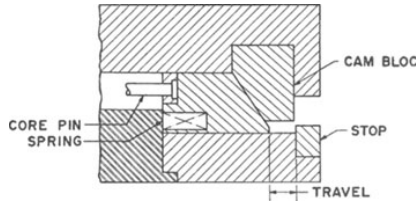


Fig. 4-83 Cam blocks used for short travel.

against the stop by spring tension. This keeps the hole in the slide in proper alignment for the angle pin when the mold is to be closed.

Cam Blocks

Another method of pulling out side core pins by using the normal movement of the press is accomplished by employing cam blocks. As the mold closes, the angle on the cam block engages the angle on the slide, forcing the slide to move in toward the plunger. The cam also acts as a lock. As the mold opens, compression springs move the slide away from the plunger, pulling the core pin. The stop limits the travel of the slide. Figure 4-83 shows the type of construction used for very short travel with the spring located in the slide. Figure 4-84 is an example used for longer travel with the compression spring mounted externally. The angle of the cam block varies from 15 to 35°.

Stripper-Plate Ejection

Stripper-plate ejection is generally used when ejector-pin marks would be objectionable on the piece parts and maximum ejection surface is required. Stripper plates are used on single- and multiple-cavity molds. An angle of approximately 5° is machined in the stripper plate and on the plunger, as shown in Figs. 4-85 and 4-86. This prevents scoring of the plunger as the stripper plate moves in and out over the plunger. The illustration shows two methods of keeping the stripper plate from coming completely off the plungers and out of the mold. The view at A shows the use of a stripper bolt to limit the travel of the stripper plate. That at B shows the return pin held to the stripper plate by a screw. This allows the stripper plate and ejector plate to operate as a unit. In more complicated designs, pull rods mounted in the stationary portion of the mold are used to activate the stripper plate.

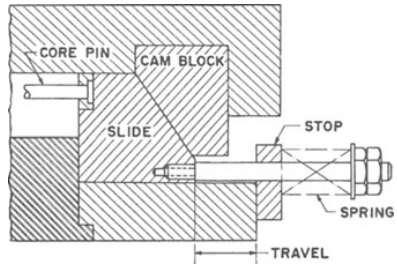


Fig. 4-84 Cam blocks used for open travel in a longer mold.

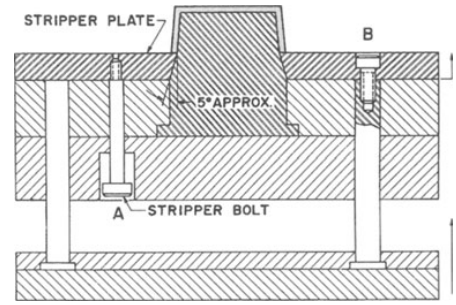


Fig. 4-85 Stripper-plate ejection in the closed position.

External-Positive-Return Systems

With large mold bases, which typically have both bigger and heavier ejector systems, Toggle-Loks by DME are used to ensure early positive ejection return. These assemblies are externally mounted on the side of the mold, as shown in Fig. 4-87. The lever is mounted to the stationary side of the mold with arms and joints located on the movable side connecting to the ejector plates. The arms and joints are positioned so that when the ejector system actuated forward, the arms pivot inward, occupying the space vacated by the retracting lever. Before the mold reaches the fully closed position, the angle on the

DNA Genotek Inc. Exhibit 2029 Page 109
Spectrum Solutions LLC v. DNA Genotek Inc. IPR2023-01424

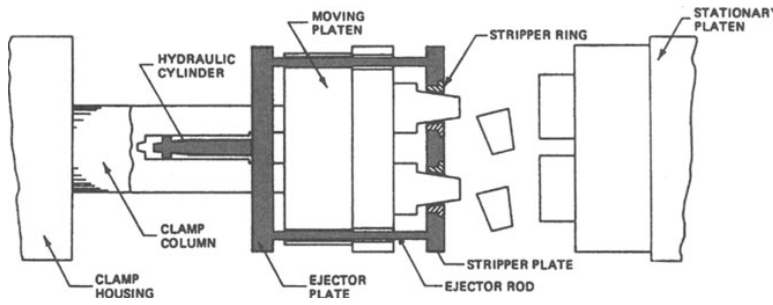


Fig. 4-86 Stripper-plate ejection in the open position.

lever contacts the radius on the pivot arm, applies pressure through the joint, and subsequently returns the ejector system to its fully retracted position.

Cam Actuation

A cam-actuated stripper plate on multicavity single-face and stack molds for lid applications is offered by Husky as an alternative to conventional mechanical linkage and hydraulically actuated methods. Unlike mechanically linked methods, the cam is designed to disengage from the rollers to prevent mold damage if the mold is accidentally opened too far. No readjustment of the cam is required before closing the mold, as it is designed to reengage automatically. The cam profile is shaped to optimize the stripper plate stroke and speed profile, resulting in reduced part hangup and increased uptime (Fig. 4-88).

The stripper plate cam mechanism consists of a simple L-shaped cam and two cam follower rollers. A spring-loaded assembly is built into the core backing plate to retract the stripper plate. Each stripper plate contains four cam mechanisms and spring assemblies to prevent cocking.

The core plate cam follower lifts the cam to begin the stripper plate's forward motion as the mold opens. The cam rollers stay in contact with the cam, ensuring smooth forward and retract motions with no "slapping" of the stripper plate to the core plate, even at maximum clamp speeds.

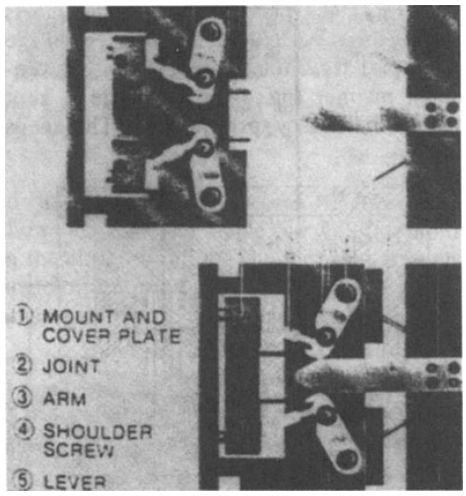


Fig. 4-87 External-positive-return system.

Sprue Bushing and Locating Ring

Figure 4-89 shows some of the dimensions found on standard sprue bushings used in injection molds. Figure 4-90 illustrates one type of locating ring. The locating ring fits over the sprue bushing and positions the mold on the platen of the press, thus aligning the injection nozzle with the sprue bushing.

Ring and Bar Ejection

The principle of the ring ejector (ring knockout) is similar to that of stripper-plate ejection. To avoid the use of a large stripper plate, each plunger is provided with an individual ejector ring. The construction shown in

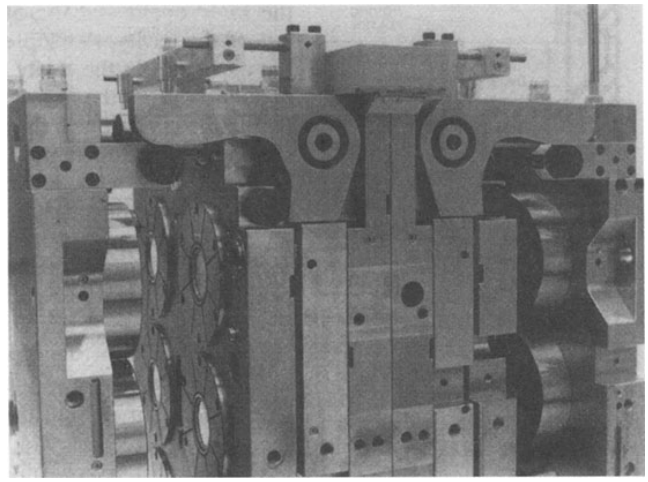


Fig. 4-88 Cam-actuated stripper plate for lid mold.

Fig. 4-91 is used to remove round piece parts from the plungers. The bar ejector, of similar construction, is used to eject piece parts from rectangular or straight walled plungers.

Top-and-Bottom Ejection

Some automatic molds are constructed with two ejector systems: a top and bottom unit. Both units ensure positive ejection of the piece parts, thus enabling the unloading devices to remove the parts mechanically. Figure 4-92 illustrates the use of the top and bottom ejection system. A spring-loaded top ejection unit is used on a six-cavity compression mold to produce the article shown in Fig. 4-92. The bottom ejector unit consists of a single ejector pin for each cavity. The top of the ejector pin is counterbored to accommodate a hexagon brass insert. As the mold opens and the plunger is moving out of the cavity, the top ejector pins hold the piece part in the cavity until the plunger is pulled out of the piece part, leaving the part in the cavity.

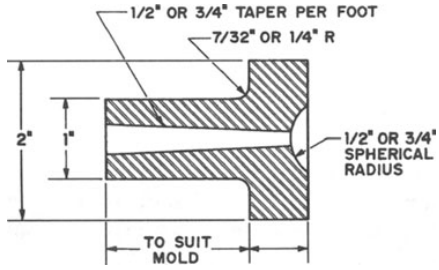


Fig. 4-89 Typical sprue bushing.

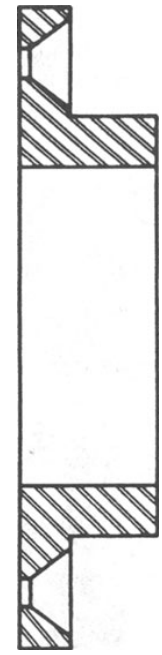


Fig. 4-90 Typical locating ring.

DNA Genotek Inc. Exhibit 2029 Page 111
Spectrum Solutions LLC v. DNA Genotek Inc. IPR2023-01424

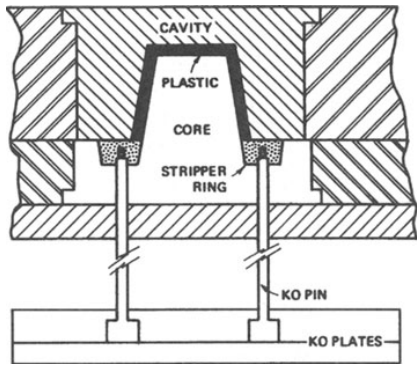


Fig. 4-91 Stripper-ring ejection.

Once the press has opened sufficiently to allow removal of the part, the bottom ejector unit is activated and the bottom ejector pin pushes on the brass insert and the piece part, removing it from the cavity. The bottom ejector pin raises the piece part out of the cavity far enough for easy removal and keeps the pin in position for easy loading of the hexagon inserts for the next cycle.

Inserts

The word "insert" has two different meanings in mold construction and molding operations. The first has to do with parts that are difficult to machine as an integral part of the cavity or plunger. These parts are machined separately from steel or the same material as the plunger or cavity and installed (inserted) in the proper position, where they become part of the molding unit. Often they are referred to as cavity or core inserts. This type

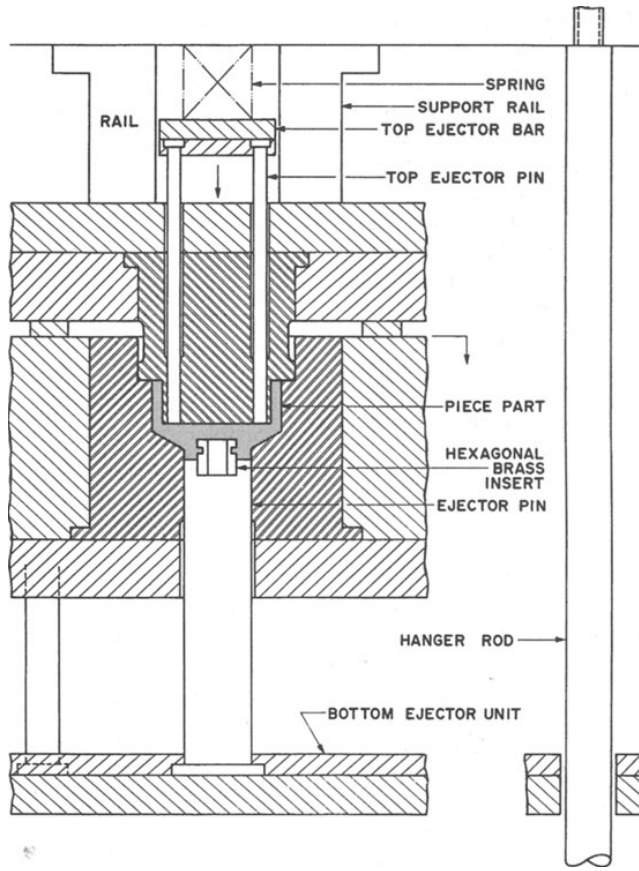


Fig. 4-92 Top-and-bottom ejection.

DNA Genotek Inc. Exhibit 2029 Page 112
Spectrum Solutions LLC v. DNA Genotek Inc. IPR2023-01424

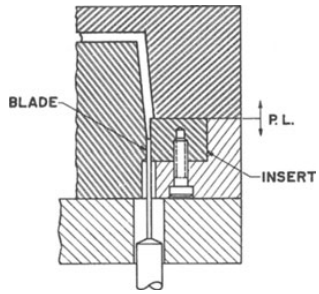


Fig. 4-93 Insert with blade ejector.

of insert is also used when the replacement of damaged or broken sections of the mold might be necessary. Sometimes, these inserts are used when minor changes in the piece part are made. They are also used in the formation of narrow slots to produce support ribs on the piece part.

The second kind of inserts are not part of the mold itself, but a separate piece, loaded (inserted) into the mold and becoming part of the finished molded piece after the material is cured or hardened around the insert in the molding process. Such inserts are used to provide greater strength in certain portions of the piece part, and to furnish external and internal threaded sections for later assembling purposes; they are also used as bearing surfaces and contacts in molding electrical connectors and subassemblies. Inserts of this type are generally made from brass, copper, aluminum, or steel. Silver and other precious metals are used for contacts in the electronics field.

Integral inserts Figure 4-93 illustrates the typical construction of an insert to mold a thin wall section below the parting line, using a blade ejector. Figure 4-94 is an example of the use of an insert in the formation of ribs on the piece part, where blade ejectors are used.

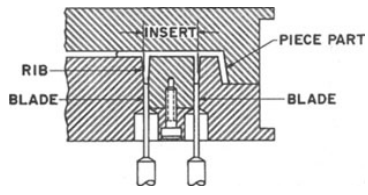


Fig. 4-94 Insert forms ribs.

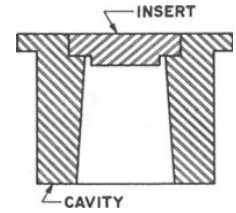


Fig. 4-95 Insert forms recess.

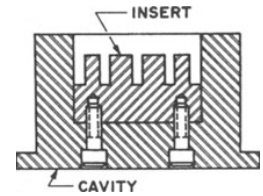


Fig. 4-96 Insert produces sharp corners.

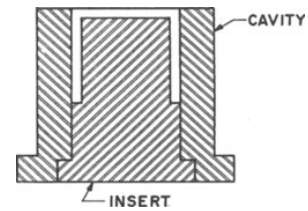


Fig. 4-97 Insert to form a long, thin wall.

Figure 4-95 is an example of an insert used to form a recess at the bottom of a deep cavity. Figure 4-96 shows how an insert is used in producing sharp corners at the cavity wall and narrow slots. Figure 4-97 shows the formation of long, thin wall sections by using an insert in the cavity. Figure 4-98 depicts an insert that contains engraving. Figure 4-99 illustrates the use of an insert for irregular detail on a round cavity or plunger. Figure 4-100 shows a typical core pin insert used in a cavity or plunger.

Loaded inserts Figure 4-101 illustrates several methods for holding inserts during

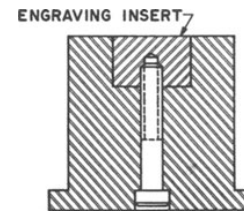


Fig. 4-98 Insert for engraving.

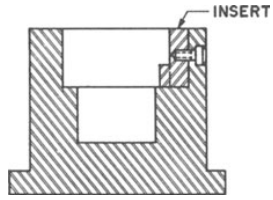


Fig. 4-99 Insert for irregular detail on round cavity.

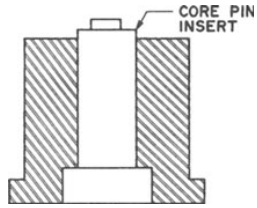


Fig. 4-100 Core pin insert.

the molding operation. Some sort of provision is made so that the insert will interlock with the material and be held firmly in the finished molded article. Common securing devices include knurling of round stock, seal grooving of projections, drilling of holes, and undercutting. The type of locking method used depends on the shape and size of the insert, and where it is located in the piece part.

Side Guide Slides

Figure 4-102 shows the sequence as sliding guides mold ballpoint pen barrels.

Ejector Blades

When the molded part has deep ribs extending into the cavity, ejector blades can be built into the bottom of the cavity. The blade push the ribs from the cavity and assure that the part will not prematurely eject from the core (Fig. 4-103).

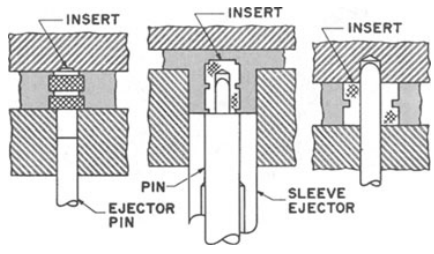


Fig. 4-101 Example of loaded inserts.

Mold Venting

Every mold contains air that must be removed or displaced as the mold is being filled with a plastic material. This air must be allowed to escape freely during injection (Fig. 4-104). At high injection speeds, insufficient mold venting may produce a considerable compression of the air, with consequent slow mold filling, premature plastic pressure buildup, and, in extreme cases, burning of the plastic (brown streaks on the molding).

Venting is done by small gaps, or *vents* (dimensions shown in Fig. 4-105) provided in the mold parting lines, or other small channels in the mold [i.e., around ejector pins (Fig. 4-106), cores, etc.]. Vents must be provided at the end of the flow path(s). A center-gated mold cavity, for instance, must be vented all around, whereas in an edge-gated cavity the vents must be provided at the point where the flow path is expected to end (generally the cavity end). In gate design, and even in article design, allowance should be made for mold venting.

Vacuum venting of molds has not yet found widespread acceptance in the injection molding of thermoplastics. However, in view of the present trend toward higher injection speeds, it is most probable that in the future, vacuum molds will be generally used to prevent venting problems.

To overcome the trapping of air or gas in a cavity, in locations that are difficult to vent effectively, molds may be designed such that all cavity vents feed into a space that is sealed from the outside of the mold (when closed) by an O-ring seal, and is connected to a vacuum reservoir through a vacuum line containing a solenoid-operated valve. In operation, as soon as the mold is closed and the transfer plunger enters the pot, the aforementioned solenoid valve is automatically opened, causing the cavities to vent rapidly into the vacuum reservoir before the molding compound has entered or filled the cavities. Two benefits

DNA Genotek Inc. Exhibit 2029 Page 114
Spectrum Solutions LLC v. DNA Genotek Inc. IPR2023-01424

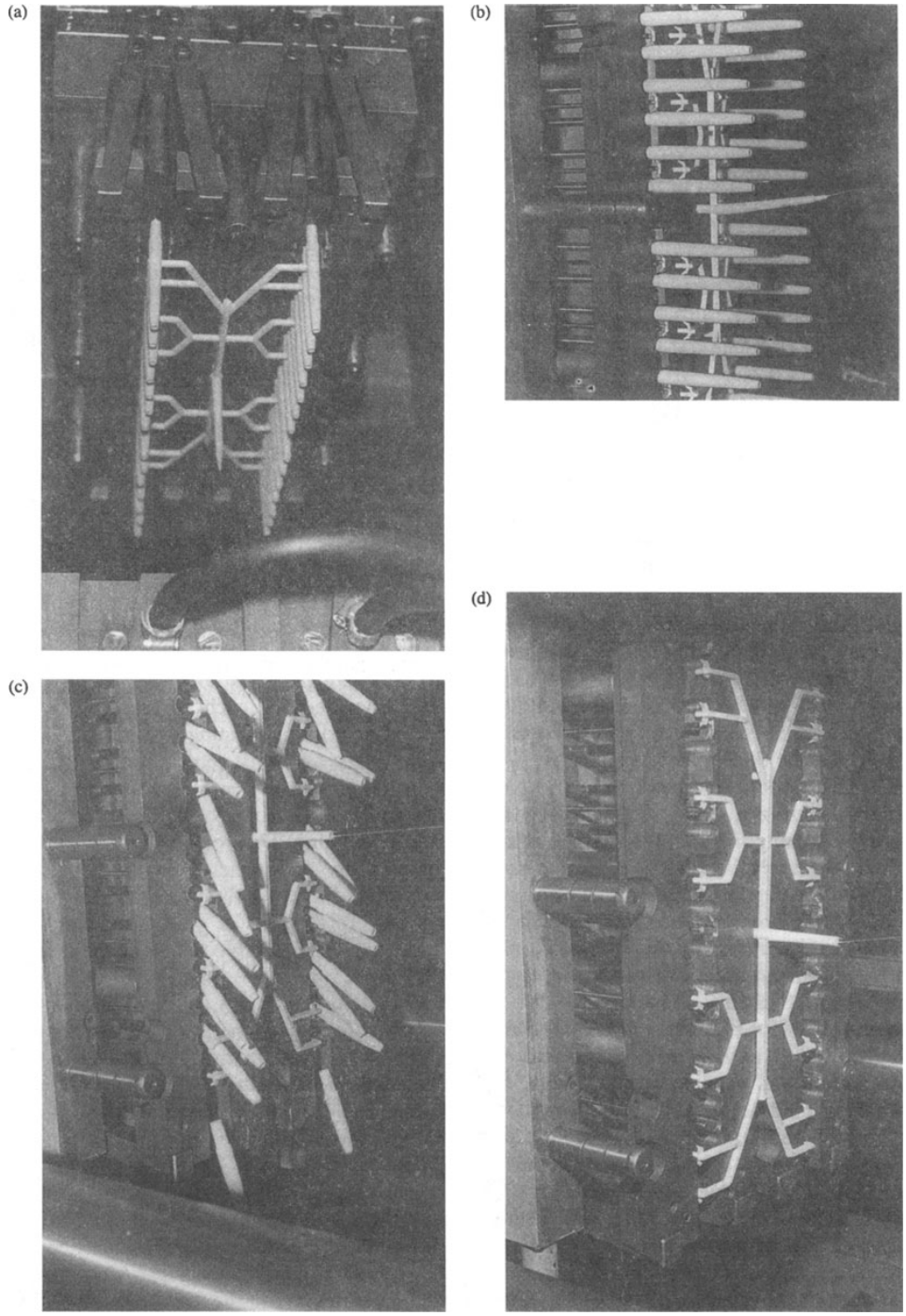


Fig. 4-102 Sequence of five views as the mold opens using sliding guides.

(e)

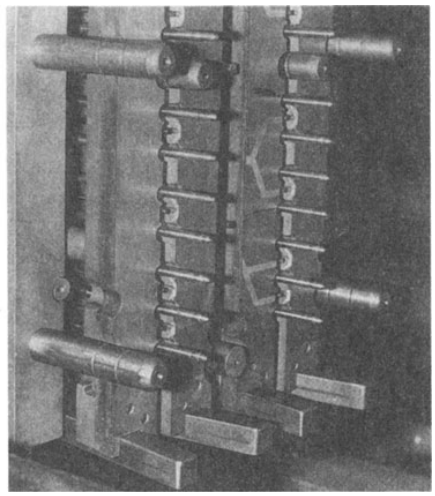


Fig. 4-102 (*Continued*)

result: First, the material, finding it unnecessary to "push" the air from the cavity through the vents, enters with a minimum of back pressure and thus fills the cavity more rapidly, leading to faster cures. Second, not only is there essentially no trapped air, and therefore no voids in the part, but such minute quantities of air as may be present are readily absorbed into solution in the molding compound because of the molding pressure.

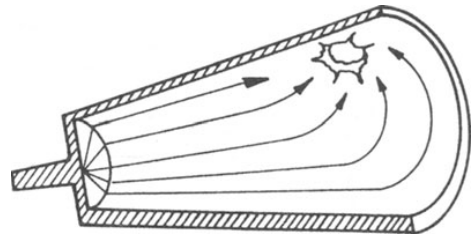


Fig. 4-104 Example of air trapped in the mold cavity.

In venting some parts, even the most minute flash may be objectionable—as with gears. Although the depth of venting specified for each material is obtained after extensive testing by suppliers of raw materials, one must remember, in addition to the measured depth, to consider the peaks and valleys from the surface roughness of machining. This roughness measurement plus the "micrometer depth" should be considered as the value indicated in the tabulation. In the case of gears and similar parts, it may be advisable to adopt the following procedure for venting: (1) vent the runner system thoroughly, (2) vent all ejector pins as indicated on the material processing data sheet, (3) water-blast mating surfaces at the parting line with 200-grit silicon carbide abrasive, and

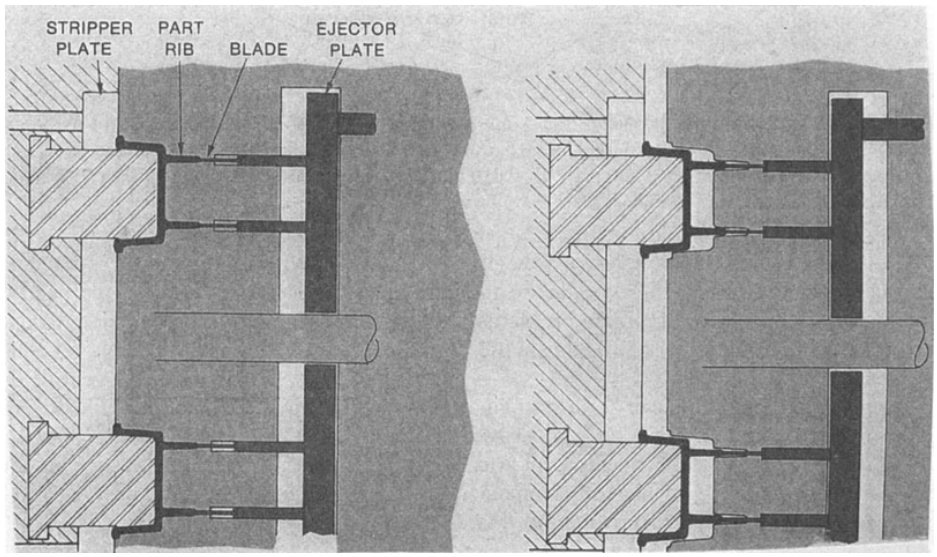


Fig. 4-103 Ejector-blade system.

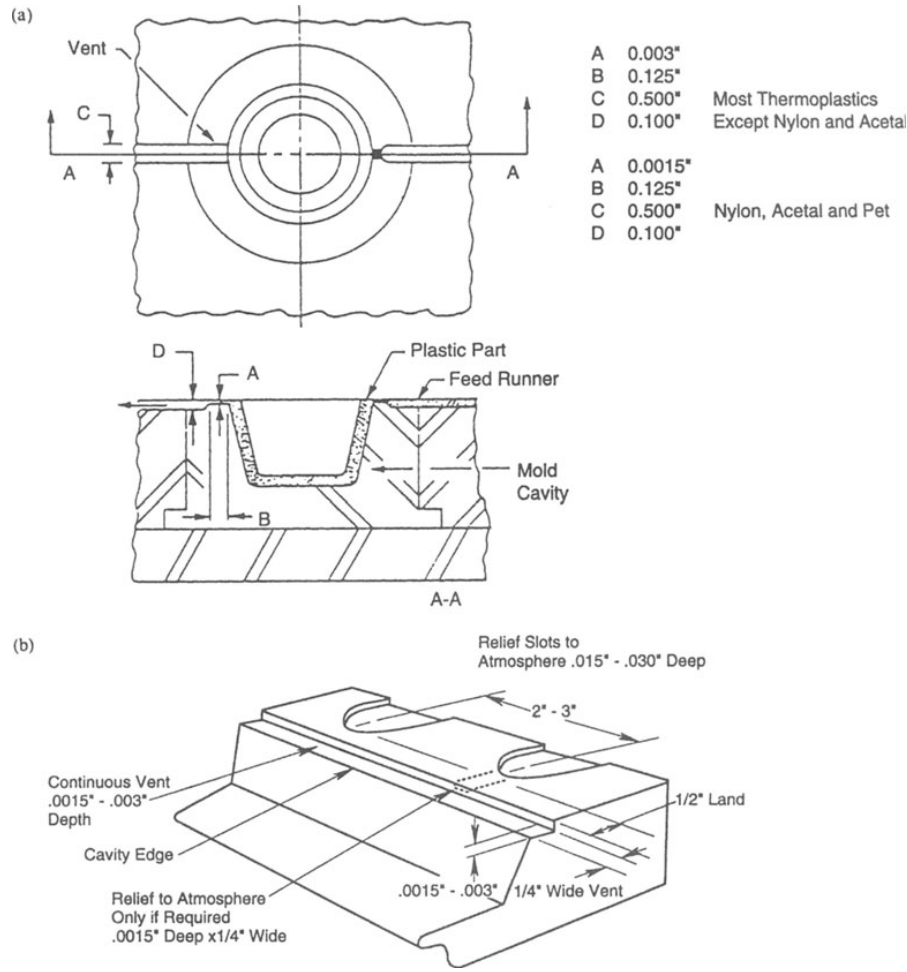


Fig. 4-105 Mold venting approaches. (a) A method of venting TPs. For most TPs except nylon and acetal, $A = 0.003$ in., $B = 0.125$ in., $C = 0.500$ in., $D = 0.010$ in. For nylon, acetal, and PET, $A = 0.0015$ in., $B = 0.125$ in., $C = 0.500$ in., $D = 0.010$ in. (b) A method used particularly with TS plastics, but also with TPs.

(4) polish the vent in the direction of flow. (It is also important to polish the cavity in the direction of melt flow—to eliminate problems on parts such as rough surface, sticking in the mold, etc.)

As an example, sticking can occur particularly in a deep container cavity. After the cavity separates from the core, the atmospheric pressure may make it difficult, if not impossible, to remove the part. To overcome this difficulty, a vent pin is used, which is held in its normally closed position by a spring (Fig. 4-107). When the material is injected, the pressure of the material on the head of the pin forces it tightly closed. When the part is to be ejected, the pin will move up when the knockout system is activated, venting the interface between the core and plastic. Additionally, air can be used to help blast the part off. In that instance, the plastic part acts like the piston of an air cylinder. Often, this provides enough force for ejection.

Vents are more important in thermosets than thermoplastics. First of all, runners

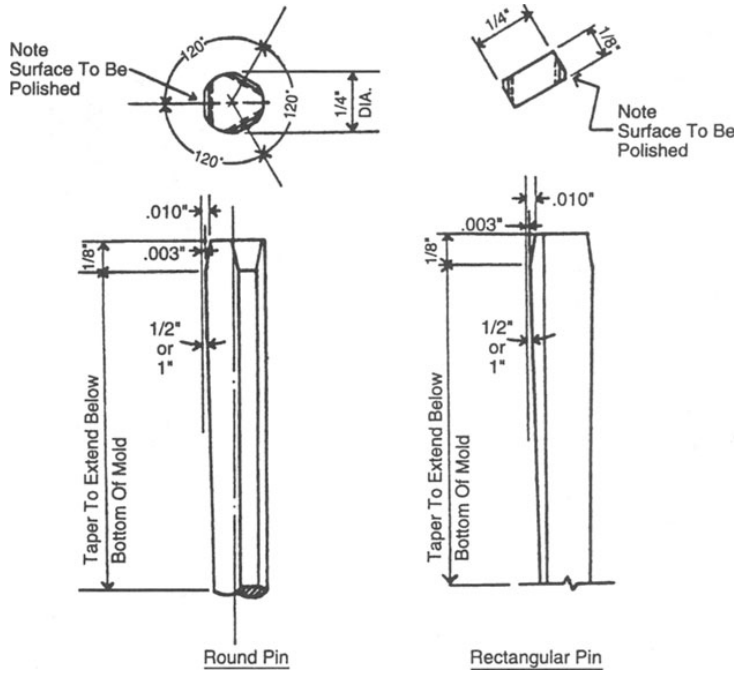


Fig. 4-106 Method of venting knockout pins.

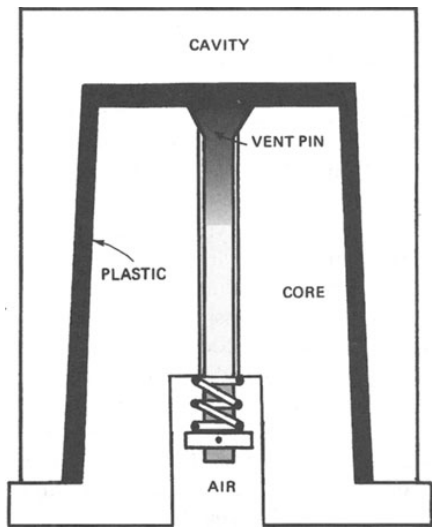


Fig. 4-107 Use of a venting pin to break the vacuum on the core. The pin is held closed by a spring; the pressure of the material on the head of the pin forces it tightly closed.

should be vented prior to approaching the gate. The vents should be the full width of the runner and 0.005 in. (0.013 cm) deep. The circumference of the cavity should be vented, and the vents should be spaced about 1 in. apart and be 0.25 in. (0.64 cm) wide and 0.003 to 0.007 in. (0.008 to 0.018 cm) deep, depending on the flow characteristics of the material.

A softer material would call for a lower value. Knockout pins should be as large as possible, and in most cases they should have 0.002-in.-(0.005-cm)-deep flats—three or four of them ground on the circumference of the diameter, with the grinding lines parallel to the length of the pins. The grind should be with a fine-grit wheel. The end of the pin should have the corner broken by 0.005 in. (0.013 cm) so that if any flash is formed, it will adhere to the part.

Occasionally, it is necessary to place knockout pins at the vent slots to ensure that the flash from the vents is physically removed, thereby assuring open vents for the following shot.

DNA Genotek Inc. Exhibit 2029 Page 118
Spectrum Solutions LLC v. DNA Genotek Inc. IPR2023-01424

Venting by Water Transfer Waterline venting is a technique based on negative-pressure coolant technology. (Mold coolant is being pulled, not pushed, through the coolant system, producing a negative pressure in the coolant system.) One easy way to vent into a waterline is through ejector pins. The only twist to this method is that the ejector pin runs through a waterline, and the molding gases vent into the coolant rather than the atmosphere. Coolant does not leak into the cavity, because it is under atmospheric pressure.

Two distinct advantages are unique to this method:

1. The pin passes right through the coolant. This prevents it from overheating, which can lead to a buildup of gummy deposits that exudes from the overheated plastic. The gummy deposits, in turn, tend to plug up the vent and make the problem even worse: The increased compression of gases further heats the pin, resulting in even more gummy deposits. If the pin is kept cool, the problem does not even begin.

2. Placement of venting pins (and just plain pins, for that matter) has one less constraint: The location of water channels no longer dictates that a pin cannot be placed there. (The reverse, of course, is also true: The location of a critical pin no longer means that a water channel cannot be put there.)

Before we leave the subject of vented ejector pins, here is another way to easily provide more venting: Vent the entire circumference of the pin. The tip of the pin is ground down to the proper vent depth and land, and then a pickup groove is ground around the base of the vent. This vents back to the atmosphere or coolant via a few large flats ground in the major outer diameter of the pins. This vent provides a great deal more venting area than a number of small flats and is easier to machine.

Porous metal provides another method of venting into the coolant. The chief advantage of this method is that a tremendous venting area is gained.

Figure 4-108 illustrates the primary concepts of the technique. It is particularly advantageous to install the vent at the top of a core such as the one shown, but it may be used in virtually any place where it can be attached in the mold and there is a waterline nearby.

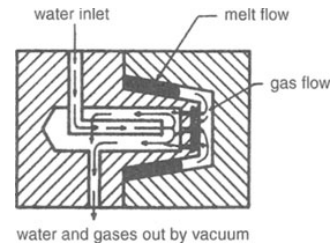


Fig. 4-108 Porous metal can be used to provide a very large venting area. The gases exhaust into the coolant. The key to the process is that the porous metal is directly cooled by the water behind it, which keeps it from overheating (porous metal has very low heat conductivity) and plugging with hot plastic. Coolant does not leak out, because it is held at subatmospheric pressure.

Pressed metal, of course, will leave a texture on the plastic, but usually it can be located either where it is not seen or where it does not matter; or it can be blended into a texture on the rest of the mold. Also, very finely woven porous metal is available that leaves only a very faint texture on the plastic.

Nor is improved venting per se the only benefit. Many new mold constructions are possible, such as gating a part, as shown in Fig. 4-108. Normally, this mold design would not be feasible because a bad turn would result at the top of the part. In this particular case, the tool would typically be designed as a three-plate mold, with its concomitant expense and complications. Many other unique designs are now possible with these techniques that reduce tool cost and improve molding efficiency and/or part appearance.

The water-transfer process was designed primarily to cool long, thin cores, such as those for pen barrels, that leave a hole in either end. The process is illustrated in Fig. 4-109. Coolant passes from one half of the mold to the other half, right through the part, when the mold is closed. When the mold is open, the supply to the mold is shut off, and both ends are subject to

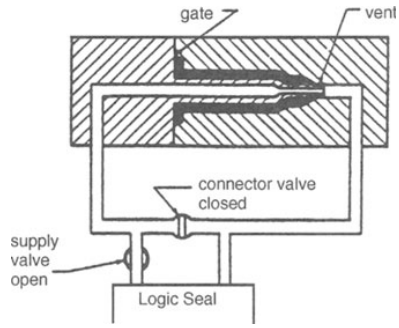


Fig. 4-109 Water-transfer mold shown closed with coolant passing through the product. Before the mold opens, both valves shift and coolant is evacuated from the mold. Venting occurs at the tip of the core.

subatmospheric pressure that evacuates the remaining coolant so that none leaks into the tool. Note that the design in Fig. 4-109 results in a parting-line vent at the end of the part opposite the gate, just where it is needed. In this case, once again, the gases vent into the coolant rather than the atmosphere.

With this type of construction, which is typical of water-transfer applications, it is in fact extremely difficult not to vent as well as cool the part, since this would require virtually perfect mating between the tip of the core and the cavity. Obviously, if venting does need to be added, it would require very little effort to grind an adequate venting clearance in the tip of the core.

Molds for Thermoset Plastics

Molds used to produce injection-molded thermoset (TS) plastic parts have been made from at least the 1940s. Although the level of technology has advanced considerably since that time, only a few specialized moldmakers have a command of these techniques. [As previously reviewed, over 90% of all injection-molded plastics process thermoplastics (TPs).] The conditions for designing molds and molding TSs are similar to those for TPs. The major difference with TSs is that after entering the mold, higher temperatures are used to solidify the plastics through chemical cross-linking (Chap. 6).

During the plasticating process, TSs initially react in a manner similar to TPs; with increasing temperature, they become more fluid. Prior to cross-linking, they pass through a low-viscosity phase. This phase normally occurs upon contact of the molding compound with the hot mold surface during the filling process. During the several seconds required for the TS to solidify as a result of the cross-linking reaction, it can penetrate cracks only a few hundredths of a millimeter in width, as if it were water. This situation is opposite to that of TPs, which solidify in less than a second upon contact with the cavity wall.

During the curing stage, TSs pass through a density minimum (a maximum in specific volume). Initially, the TS expands with increasing temperature and then shrinks as the main reaction starts. The result can be that during the holding pressure phase, TS is forced into the parting line, cracks in the mold cavity, or even back into the sprue bushing at a pressure considerably above the maximum injection pressure.

All TSs generally contain a high percentage of fillers. Depending on the type of filler (glass fiber, calcium carbonate, wood flour, etc.) and the amount included, melt flow properties significantly change, as well as shrinkage and warpage of the molded part. Also, many TSs release gases (water vapor, formaldehyde, etc.) such as the phenolics. (TS polyesters do not release these volatiles.) The gases can interfere with the filling of cavities and part performance.

Mold Construction

As with the different TPs, there are rules and guidelines to follow when working with TSs. Some of these conditions will be reviewed. As an example, in order to reduce flash that unavoidably forms when processing TSs, the mold must be constructed to be extremely rigid. All fits in the vicinity of cavities must have very close tolerances (clearance around ejectors of 0.01 to 0.02 mm). Flash only a few hundredths of a millimeter in thickness can readily be removed by means of blasting in automatic deflashing machines,

whereas flash having a thickness of a few tenths of a millimeter requires manual trimming or machining.

Thermoset parts are not yet fully cured upon ejection and are still brittle. It is thus necessary to use a large number of ejector pins in order to prevent damage to the part upon ejection. Ejectors are required especially behind all ribs, bosses, and the like. An old rule of thumb says that a thermoset mold requires twice as many ejectors as a thermoplastic one.

Guides for the injection pressure to be used in determining the IMM clamping force are available from material suppliers. As an example, phenolic, amino, and epoxy plastics have a cavity pressure of 300 to 400 bar (4,350 to 5,800 psi) and flash land pressure of 500 to 600 bar (7,250 to 8,700 psi). With TS (unsaturated) polyester plastics, the cavity pressure is 100 to 300 bar (1,450 to 4,350 psi), and the flash land pressure 400 to 500 bar (5,800 to 7,250 psi). The low values apply to heavy-walled parts and for short flow paths in molds with flash lands. (Flash lands are regions around the cavities of molds.)

The installed heating capacity should be 20 to 30 W/kg of mold weight in order to achieve an acceptance heatup time and provide for stable temperature control. The heaters should be distributed uniformly throughout the mold. When using electric resistance heaters, placement of the heating circuits and heaters should be thoroughly investigated, for example, by means of a computer simulation. In large molds, between 8 and 16 control circuits are used today. Sealed high-performance rod heaters should be considered state of the art.

In order to permit stable temperature control, thermocouples should be located a distance of 12 to 15 mm from heater rod wells. They must not, however, extend to within the immediate vicinity of the mold cavities in order that the cyclic temperature variations at the cavity surface do not interfere with the controller.

For economic reasons, large molds are often heated with steam, since this form of energy is lower in cost than electricity. Thermoset molds should be thermally insulated all around to the greatest extent possible in order to reduce energy consumption and prevent heating of the molding machine.

Cold-Runner Systems

In the past decade, the cold-runner technique has advanced significantly and is finding a wide field of applications. As previously mentioned (see the section entitled "Runner Systems"), with TS cold runners the amount of plastic lost in the form of the sprue and runner, as well as the cycle time, can be reduced. This is particularly true with multiple-cavity molds. Cold runners are generally used with easy-flowing TSs (phenolics, polyesters, and aminos).

Injection–Compression Moldings

This technique, also called coining, is a very economical process for molding TS parts. Very high-quality parts can be produced automatically. See the section on Injection–Compression Molding (Coining) in Chap. 15.

Mold Cooling

Overview

Controlled cooling channels are essential in a mold for TPs and require special attention in mold design. The cooling medium must be in turbulent flow, rather than laminar flow, in order to transfer heat out of the molded part at an adequate rate. The coolant is usually water, but can be any liquid or gas (such as air) that can absorb heat and transfer it efficiently away from its source. Coolants are used in molds, chillers, etc. Water is one of the most effective and low-cost coolants. It may be mixed with an antifreeze such as ethylene glycol for operation below the freezing point.

Channels (passageways) are located within the body of the mold through which the cooling and/or heating medium can be circulated. With TS plastics, heating is required and can be accomplished by circulating steam, hot oil,

or other heated fluid. However, the usual method involves inserting electrical heating elements (calrods) or probes in the mold rather than using the channels.

A laminar (nonturbulent) flow is not desirable in a coolant system, which requires a nonlaminar flow so that the fluid moves in all different directions in the mold cooling channels. With turbulence, more heat will be removed, since as the fluid on the inside surface of the channel is heated, it moves away and is replaced by cooler fluid. With laminar flow the hot fluid would build up on the wall and act as an insulator.

Baffles can be used in the water channels to divert or restrict the flow to a desired path. They also aid in developing turbulence. These cooling baffles (ribs, plugs, etc.) provide more uniform cooling action within the mold. Other devices for that purpose include *bubblers* (channel space). Inserted into a mold cavity, they facilitate water flow.

Reynolds number The Reynolds number is used to determine whether the coolant will be turbulence. Also called N_{re} or Damkihler number V (DaV), it is a dimensionless number that is significant in the design of any system in which the effect of viscosity is important in controlling the velocities or the flow pattern of a fluid. It is equal to the density of a fluid, times its velocity, times a characteristic length, divided by the fluid viscosity. This ratio is used to determine whether the flow of a fluid through a channel or passage, such as in a mold, is laminar (streamlined) or turbulent.

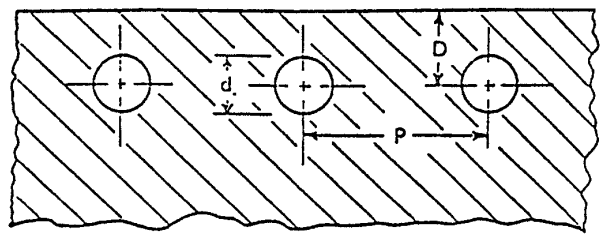
Design Considerations

One of the most important aspects of mold design is the provision of suitable and adequate cooling arrangements. In all injection molding, even though it may involve a heated mold, an essential purpose of the mold is to cool the molten plastic. If a mold had no means of cooling and was insulated to prevent any escape of heat by conduction, convection, or radiation, it would quickly reach the temperature of the material being molded and no longer fulfill its function.

The cooling system is an essential mold feature, requiring special attention in mold design. It should ensure rapid and uniform cooling of the molding. In the design of mold components and the layout of guides and ejectors, allowances should be made for the proper size and positioning of the cooling system.

Rapid cooling improves process economics, whereas uniform cooling improves product quality by preventing differential shrinkage, internal stresses, and mold release problems, as well as shortening the molding cycle. Rapid and uniform cooling is achieved by a sufficient number of properly located cooling channels. The location of these channels should be consistent with the shape of the molding and as close to the cavity wall as allowed by the strength and rigidity of the mold.

Increasing the depth of the cooling lines from the molding surface reduces the heat-transfer efficiency, and too wide a pitch gives a nonuniform mold surface temperature (Figs. 4-110 and 4-111). Straight-drilled lines



Recommended depth and pitch
d = Diameter of Water Line = 7/16 to 9/16 inch
D= Depth of Water Line = d to 2d
P = Pitch = 3d to 5d

Fig. 4-110 Recommended depth and pitch of mold cooling channels. The depth D should be of 1 diameter for steel, $1\frac{1}{2}$ for beryllium copper, and 2 for aluminum.

DNA Genotek Inc. Exhibit 2029 Page 122
Spectrum Solutions LLC v. DNA Genotek Inc. IPR2023-01424

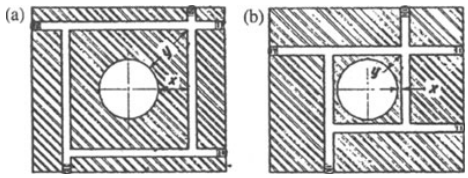


Fig. 4-111 Guide for channeling. (a) Proper channeling: $x > \frac{1}{2} y$ represents the shortest distance from channel to cavity and y the greatest distance from channel to cavity. (b) Improper channeling: $x < \frac{1}{2} y$. Here x is the shortest distance from channel to cavity, and y the greatest distance from channel to cavity.

are preferred to bubblers. When it is necessary to use bubblers, they should be designed so that the cross-sectional area remains constant for the entire circuit. For tube bubblers, areas on both sides of the tube should be equal. Materials with higher thermal conductivity should be used if all the heat cannot be removed with a steel mold. Examples of layouts usually employed for cooling systems are shown in Fig. 4-112.

The desired location of the heating–cooling passages is in the mold inserts themselves; they should be located close to where most of the heat has to be dissipated—that is, where most of the material is located.

The inclusion of fluid passages in the A and B plates, as well as in their supporting plates, adds to the ability to control cavity temperature, but not to the extent that one might expect. Because steel surfaces always have some heat-insulating film, the contact between them is never such as to induce the best conductivity. This has been verified in practice by interposing a sheet of soft copper or brass between the B plate and its supporting plate on the core side, and checking for temperature while all other conditions remained the same. The average drop in temperature was found to be 25°F (14°C), and the core came close to the temperature of the

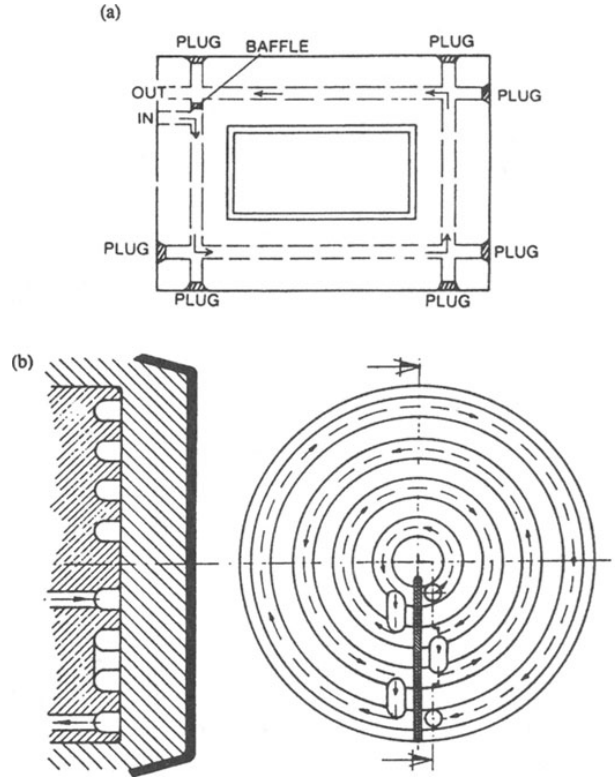


Fig. 4-112 Examples of cavity cooling systems.

cavity. Prior to this change, the core was running considerably hotter than the cavity; this made it possible to reduce the cycle time by 30%. When a core consists of numerous thin sections that are difficult to arrange for individual control, the addition of a good heat conductor between plates may accomplish the desired result, provided that there are enough passages in the plates to make good heat exchange possible.

Fluid passages for effective mold and part cooling should be placed to cover most of the molding surface and to be close to the mold face. However, the distance between the mold face and fluid passage opening has to be large enough to resist distortion or flexing of the metal under injection pressures. The inlets and outlets for each cavity should be connected in parallel to their source of supply, thereby ensuring uniform heat transfer. The dimensions of the fluid passages should be such so as to create a turbulent flow, since turbulent flow will dissipate about three times as many Btu's per hour as laminar flow.

Assuming that a satisfactory cavity cooling–heating system has been provided, we now have to concern ourselves with thermally isolating the mold insert from the mold base. One practical way to do so is to provide circulating passages in the supporting plates and to maintain temperatures in them that will enable the cavity inserts to perform their function properly (i.e., consistently dissipate the heat introduced by the molded part).

Even though the specification is for straight, smooth, and dimensionally correct openings, the cores making them will require special attention to temperature control. The nature of cores is such that material shrinking over them produces intimate contact, and the bulk of the heat from the plastic is conducted into them. This condition necessitates an efficient way for dissipating the heat from the cores.

Cavity and core temperature control is also important to the proper functioning of the mold base. If cores are permitted to exceed the temperature of the cavities, the heat of the cores will ultimately transfer to the plate containing them. The B plates also hold the bushings for the leader pins. It frequently occurs that there is a difference of 30°F (17°C) between mold halves. What would this mean for a 24-in. (61-cm) mold base? The expansion of the hotter side will be

$$\begin{aligned}\text{Expansion} = {} & \text{coeff. of linear expansion} \\ & \times \text{length of mold} \\ & \times \text{temperature difference}\end{aligned}$$

Using a handbook, we find the coefficient of linear expansion for steel. Substituting, we have

$$\begin{aligned}\text{Expansion} &= 6.33 \times 10^{-6} \times 24 \text{ in.} \times 30°\text{F} \\ &= 0.0046 \text{ in. } (0.0117 \text{ cm})\end{aligned}$$

This expansion will cause binding, misalignment, difficulty in mold opening and closing, and, in the long run, excessive wear on the components that work together.

Another mold component that is affected by the temperature of the mold halves at the parting line is the stripper plate. In most cases, the stripper plate is near—or at—room temperature. For the majority of cases, the temperature difference in the stripper plate is compensated for by having an adequate clearance between plate, pin periphery, and pinhead to allow them to move freely to whatever position conditions dictate. There are cases, however, in which this clearance provision does not apply. For example, when sleeve ejection is needed and the core over which the sleeve slides is attached to the rear clamping plate, temperature control of the clamping plate and stripper plate becomes a necessity. Another way to approximate the desired condition is to insulate the clamping plate with a material such as transite board [about 0.5 in. (1.27 cm) thick], and let the steel of the base absorb enough heat to permit free working of the sleeves over the core pins.

It is best to calculate the elongation of plates under the particular condition and to decide on the basis of the calculation what action should be taken.

Basic Principles of Heat Flow

Heat flows from a body of higher temperature to one of a lower temperature. It is the

DNA Genotek Inc. Exhibit 2029 Page 124
Spectrum Solutions LLC v. DNA Genotek Inc. IPR2023-01424

temperature difference—and not the temperatures themselves—that determines the flow of heat. The greater the difference in temperatures between two bodies, the greater the rate of heat flow between them. Heat can be transferred from one medium to another in three ways: radiation, conduction, and convection.

In the molding process of thermoplastics, we are not concerned with radiation heat, except for some postmolding operations involving stress relieving, straightening, or similar operations. In those operations, some source of infrared heat is utilized for the rapid change of temperature of defined areas.

Conduction of heat is of vital importance to the molding operation. The mold material performs the job of conducting the heat from the plastic to the mold and its circulating lines. For a short time—that is, while the plastic is solidifying—the heat is flowing from the plastic material through the mold; from there, it is carried away by a circulating medium. After temperature equilibrium is attained, heat flows in the reverse direction in order to maintain the temperature of the molded part at the desired level.

Extensive tests of various substances have led to verification of a formula for calculating the heat to be transferred by conduction in order to maintain certain temperature conditions (2). This formula is

$$H = \frac{KAT(t_2 - t_3)}{L}$$

where A = area of the cavity in contact with the molding material
T = time, in hours, from the instant the plastic enters the cavity until the time ejection starts (that is, cycle time per shot)
t_2 = temperature of the injection plastic
t_1 = temperature of the circulating medium
L = distance from the face of the mold to the start of the hole in which the circulation of the medium takes place

The quantity H in the above formula is the heat in Btu conducted through a substance (mold) with a surface area A in square feet, during a time T in hours, when the difference in temperature is $t_2 - t_1$ in °F, the length or thickness through the substance (mold) is L in feet, and K is the thermal conductivity of the substance, expressed in Btu per hour, per square foot, per °F, per foot of length.

Table 4-11 Values of the thermal conductivity K for different materials at 212°F (100°C)

	K(Btu/ft-°F–h)		
Metals		Other Materials	
Stainless steel	10	Polystyrene	0.07
Tool steel (H 13)	12	Polypropylene	0.07
Tool steel (P-20)	21	Air	0.14
Beryllium copper	62	Nylon	0.14
Kirksite	62	Polyethylene	0.18
Brass (60-40)	70	Water	0.39
Aluminum	100		
Copper (pure)	222		

The conductivity K, which is related to the molding conditions, changes in particular with the temperature at which conduction takes place. Most of the materials of interest to the plastics processor have been tested at 100°C (212°F). Values for K at that temperature are listed in Table 4-11.

The value of K might be viewed as a consideration for cavity material selection, but in most cases the selection is made for performance over a large number of pieces, integrity of shape, and controllability of fabrication, and the resulting value of K must be accepted. However, one must keep the K of the selected material in mind while designing the heat-transfer system. The cooling arrangement will be more elaborate for a material with a low K value than for a material of higher K.

The K value can be used most advantageously for small-size deep cores where a straight and uniform opening is needed. For this application, it can be calculated whether a beryllium copper pin or a steel pin with a copper core will be more effective in conducting the heat away. The area exposed to the plastic material times the K value will provide a comparative figure for the preferred selection, since all other factors are common.

DNA Genotek Inc. Exhibit 2029 Page 125
Spectrum Solutions LLC v. DNA Genotek Inc. IPR2023-01424

For example, a $\frac{3}{8}$-in.-(0.95-cm)-diameter pin in a 1.5-in.-(3.81-cm)-deep cored hole will have for beryllium copper a value of

$$\text{Area} \times K = \tfrac{3}{8} \times 3.14 \times 1.5 \times 62 = 109$$

For a steel pin with a $\frac{5}{16}$-in.-(0.79-cm)-diameter copper insert, the corresponding value is

$$\tfrac{5}{16} \times 3.14 \times 1.5 \times 222 = 326$$

The relative heat conduction using beryllium copper, 109, against that using the steel pin inserted with a copper rod, 326, points to the decided advantage of the latter. It is assumed that during insertion of the copper, intimate contact with the steel is established.

The distance L to the circulating-fluid opening is determined by strength considerations, namely, by the need to limit the deflection and by the possibility of thermal fatigue. L can be calculated by viewing the situation shown in Fig. 4-113 as a beam fixed at both ends with a load in the middle. Using a handbook (under "Beams, Stresses in"), we find that the stress in the middle is

$$S = \frac{Wl}{8Z} = \frac{WD}{8Z}$$

where W = load on 1 sq in. of hole opening
= 20,000 psi (1,378 MPs)
l = length of beam = D = 0.4375
Z = section modulus

$$= \frac{bd^2}{6} = \frac{bL^2}{6} = \frac{2.29L^2}{6}$$

$$b = \frac{1}{0.4375} = 2.29$$

$d = L$
S = safe load stress = 10,000 psi

$$Z = \frac{WD}{8S} = \frac{20{,}000 \times 0.4375}{8 \times 10{,}000}$$

$$= 0.1094$$

Substituting 0.1094 for Z, we have

$$0.1094 = \frac{2.29L^2}{6}$$

$$L^2 = 0.2862$$

$$L = 0.535$$

Most circulating holes are many inches long, and so the chances of a drill "running out" exist; therefore, it would be more practical to take the distance L as $\frac{9}{16}$ to $\frac{5}{8}$ in. (1.43 to 1.59 cm) for H-type steels and $\frac{3}{4}$ in. for steels that are not formulated for thermal fatigue.

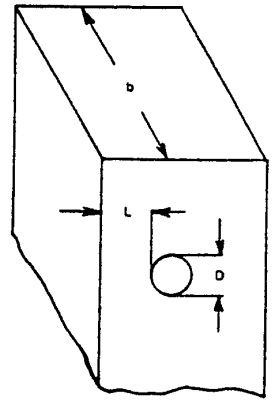


Fig. 4-113 Coolant-hole distance from molding surface.

The next step is to take account of the heat transfer by convection. Heat transfer by convection takes place when a medium, be it liquid or gas, moves from one place to another and carries heat with it; thus, the water or synthetic oil moved by a mold circulator through the circulating passages in the mold convects heat for maintaining a desired temperature. Heat is taken on or given off by the circulating medium according to the basic heat equation

$$H = MS(T_2 - T_1)T$$

Here H is the heat content in Btu, and M is the weight of material that circulates in time T hours and produces a temperature difference of $T_2 - T_1$ in °F between the inlet and outlet of the mold, respectively. S is the specific heat of the circulating medium. T is the duration of the curing cycle. This basic heat formula can be expressed in terms related to mold design requirements.

Substituting the appropriate values for M, we have

M = volume × specific gravity
volume = Q = area of passage × velocity
× duration of flow (curing cycle)
= $A \times V \times T_f$

where $A = 0.7854 \times [(\frac{11}{32})^2, (\frac{7}{16})^2, (\frac{19}{32})^2]$
$= [0.073, 0.118, 0.217]$ sq in.
V = velocity of fluid in ft/sec
T_f = time in hours during curing

Thus

$$M = A \times V \times T_f \times \text{specific gravity}$$

For a given passage opening in the mold and given cycle time, the only element that is controllable is the velocity of the circulating fluid. It has been established that within a certain range of velocities, the heat transfer is decidedly improved. This is the range in which turbulent flow takes place as contrasted with predominantly laminar flow. Turbulent flow is most effective in heat transfer because of the transverse movement of the liquid particles. In laminar flow, the liquid particles arrange themselves in parallel layers with velocities at their highest in the center and decreasing in parabolic shape as they approach the wall of the passage; thus, the layer next to the wall is moving very slowly, with little capacity to pick up heat.

Reynolds number The flow conditions in passages, whether laminar or turbulent, are characterized by a ratio known as the Reynolds number. The formula for the Reynolds number is

$$R = \frac{7{,}740VD}{n} \text{ or } \frac{3{,}160Q}{Dn}$$

where V = fluid velocity, ft/sec
D = diameter of passage, in.
n = kinematic viscosity, centistokes
= 1.3 for water at 50°F (see Table 4-12)
Q = flow rates, gpm (cu. m/min)

A Reynold's number of 2,000 or less yields laminar flow; turbulence sets in at values of 3,500 to 5,500 or even higher. Between 2,000 and 3,500, there is a transition stage. Viscosity appears in the formula for the Reynold's number, and because viscosity changes with temperature, we have to relate the flow to a specific water temperature to establish the flow rate and range of Reynold numbers. For 50°F (10°C) entrance water and a $\frac{7}{16}$-in.-(1.11-cm)-diameter opening, the minimum flow rate for turbulent flow will be

Table 4-12 Kinematic viscosity of water

Water Temperature (°F)	Kinematic Viscosity (cS)
32	1.79
50	1.30
68.4	1.00
100	0.68
150	0.43
212	0.28

$$R = 3{,}500 = \frac{3{,}160Q}{0.4375 \times 1.3}$$

$$Q_{\min} = \frac{3{,}500 \times 0.4375 \times 1.3}{3{,}160}$$

$$= 0.63 \text{ gpm } (0.0024 \text{ cu m/min})$$

For a Reynolds number of 5,500, we will have an average flow rate of

$$Q_{\text{avg}} = \frac{5{,}500 \times 0.4375 \times 1.3}{3{,}160}$$

$$= 1.00 \text{ gpm } (0.0038 \text{ cu m/min})$$

By substituting in the formula the $\frac{1}{8}$- or $\frac{3}{8}$-in. (0.32- or 0.95-cm) pipe hole size, we can figure the corresponding flows. For $\frac{1}{8}$-in. (0.32-cm) pipe hole size [$\frac{11}{32}$-in. (0.87-cm) diameter], the values of flow rate are

$$Q_{\min} = 0.5 \text{ gpm}$$
$$Q_{\text{avg}} = 0.75 \text{ gpm}$$

For $\frac{3}{8}$-in. (0.95-cm) pipe hole size [$\frac{19}{32}$-in. (1.51-cm) diameter], the values are

$$Q_{\min} = 0.855 \text{ gpm}$$
$$Q_{\text{avg}} = 1.34 \text{ gpm}$$

At a Reynolds number of 3,500, the heat conduction is 1.5 times that for laminar flow; at 5,500, it is almost 3 times better.

For water temperatures above 50°F (10°C), the corresponding viscosity will be substituted in the formula, and a new flow rate will be obtained.

When the flow in each line approaches the values shown, a decided improvement in heat conduction will result.

DNA Genotek Inc. Exhibit 2029 Page 127
Spectrum Solutions LLC v. DNA Genotek Inc. IPR2023-01424

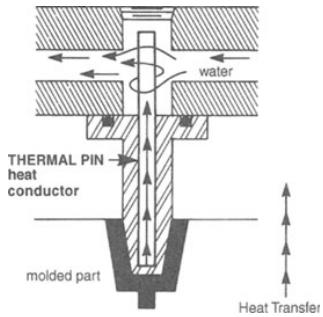


Fig. 4-114 Schematic of a heat pipe.

Heat Transfer by Heat Pipes

A heat pipe is a means of heat transfer that is capable of transmitting thermal energy at near-isothermal conditions and near-sonic velocity. The heat pipe consists of a tabular structure closed at both ends and containing a working fluid (Fig. 4-114). For heat to be transferred from one end of the structure to the other, the working liquid is vaporized; the vapors travel to and condense at the opposite end, and the condensate returns to the working liquid at the other end of the pipe. The heat-transfer ability of saturated vapor is many times greater than that of solid metallic material.

Heat pipes can be used to either remove or add heat. The smaller heat pipes, which can be used to operate against gravity, are equipped with thick homogeneous wicks and have higher thermal resistance, so that the heat transfer will not be quite as fast as in the case of gravity-positioned pipes. Even with the higher thermal resistance, these heat pipes still have a very high heat-transfer rate in comparison with solid metals.

Heat Balance of Halves

Some products are so shaped that the heat from the plastics is equally absorbed by each mold half. The vast majority of parts, however, have a core of some depth and a cavity that surrounds the core. In this type of mold, the heat absorption of each half is different.

Mold Connection for Fluid

Mold temperature connections should be placed away from the operator side and recessed wherever feasible so that danger of damaging them is eliminated. Whenever quick-disconnect couplings are used, care should be taken to see that the openings in the fittings will not restrict the flow to the mold and to ensure that the proper velocity for turbulent flow is maintained.

Cooling Time

In addition to mold, raw material, and machine costs, the cost of injection molded articles depends on the molding cycle. A large part of the cycle is accounted for by the time required to cool the molding to mold release temperature. This time depends on the heat of the molding.

In principle, the molding may be released from the mold as soon as its outer layer is sufficiently rigid, at a temperature called the mold release temperature. The inside of the molding will often still be considerably hotter than the outer part. Minimum cooling time required to reach mold release temperature is governed by:

1. Wall thickness of the molding
2. Difference between polymer and mold temperatures
3. Difference between mold release temperature of the article and mold temperature

The minimum cooling time may be estimated from the following equation (4):

$$S = \frac{-t^2}{2\pi\alpha} \log_e \left[\frac{\pi}{4} \frac{(T_r - T_m)}{(T_c - T_m)} \right]$$

where S = minimum cooling time (sec)
t = thickness of molding (in. or cm)
α = thermal diffusivity of material (sq in./sec or sq cm/sec)
T_r = ejection temperature of molding (often the heat distortion temperature is used) (°F or °C)

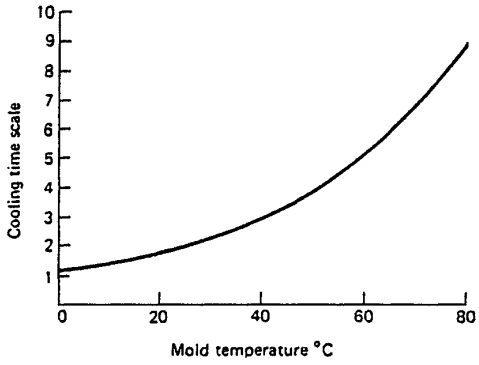


Fig. 4-115 Effect of mold temperature on cooling time.

T_m = mold temperature (°F or °C)
T_c = cylinder temperature (°F or °C)

Based on this formula, Fig. 4-115 illustrates the effect of mold temperature on cooling time.

Cooling with Melt Pulses

Rather than having the coolant continually running, one may have it flow only after the melt fills the cavity(s). Solenoid-controlled valves placed in the incoming coolant line are used to open or shut off the coolant. Thus when the melt enters the mold, it is not subjected to a fast cooling shock.

According to proponents of pulse cooling, conventional continuous-coolant-flow mold temperature control generates stable temperature gradients and thus stable isotherms (surface of constant temperature in the cooling channels). Although heat travels quickly from the plastic to the metal to the coolant, the mass of steel between the channels is under utilized for heat absorption. Pulse cooling, because it is not continuous, eliminates the steady isotherms that segregate a conventional cooled mold. Heat from the part is absorbed not only by the cooling channels, but also by the large mass of steel on the shop side of the mold. When the fill stage is complete, coolant circulates quickly, removing excess heat and quickly bringing the mold and part back to minimum temperature. This cyclic process is targeted to reduce cycle time 6 to 10%, lessen molded-in stress, reduce scrap, and reduce energy use.

Flood Cooling

This system, used particularly in blow-molding molds, involves internal flooding in a confined open chest that surrounds the mold cavity, rather than using drilled holes (Fig. 4-116) (Chap. 15). However, drilled holes can also be used or combined with the flooding action.

Spiral Cooling

In this method of mold cooling, the cooling medium flows through a spiral cavity in the body of the mold.

Cooling Rates

The plastic melt cooling rate is usually the final control among the variables associated with the final plastic product performance. This variable influences factors such as melt flow rate, residual stress, and degree of orientation. Appropriate heating and cooling rates for amorphous and crystalline plastics differ. If they are not properly controlled, product performance will be suboptimal or even unacceptable.

Cooling Temperatures

Lowering coolant temperature below the required level is often supposed to speed up heat removal; actually, the reverse is often true. Lowering the temperature reduces water chiller capacity. If possible, avoid temperatures below 40°F (4°C), since at such temperature ethylene glycol (EG) has to be added to the water to prevent freezing. Going lower requires more EG, making the solution more viscous, increasing the required pumping power, and thus increasing the operating cost.

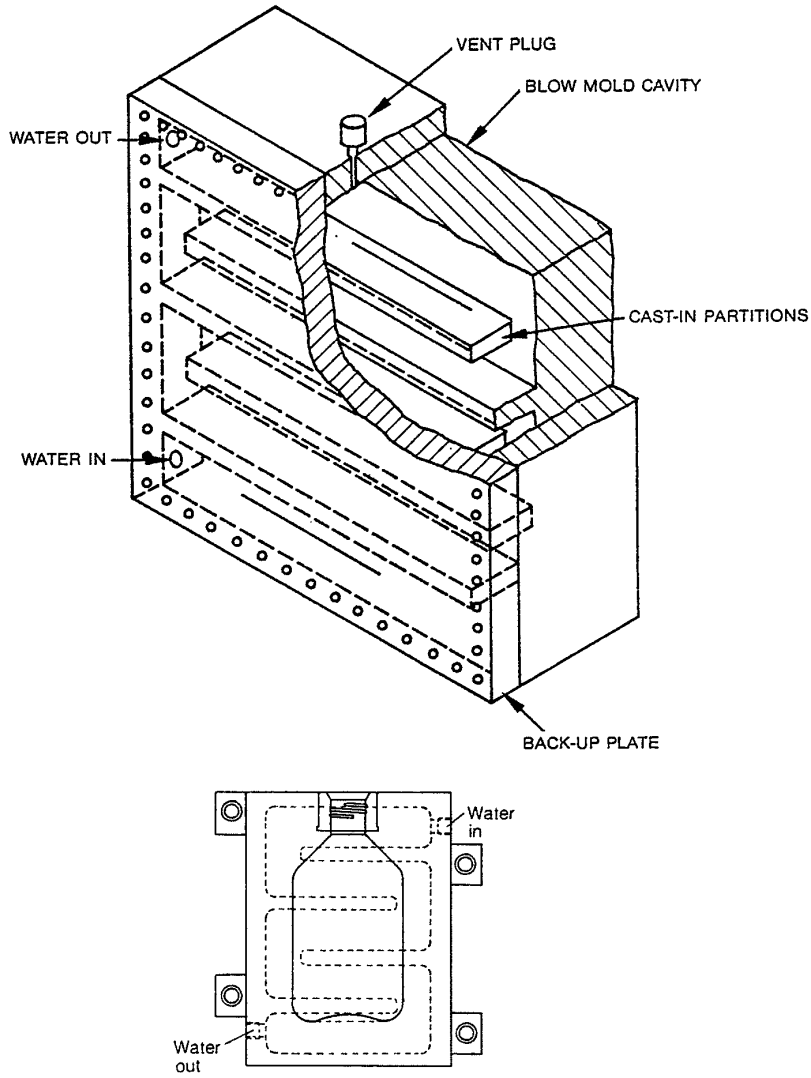


Fig. 4-116 Flood cooling: a baffle cooling system used in blow molding.

Cooling Flow Meters

Devices can be put inline with the supply and return water to measure the temperature, pressure, and flow rate of water through the mold. As an example, the *rotometer* is a water flow meter that can be installed in the water line. Flow is through a vertical transparent tube marked with a scale. A ball-shaped float (or other device) is inside the tube. It moves up and down according to the water flow rate. (There are also airflow cooling meters to monitor airflow around the molds.)

Undercuts

Ordinarily, when the mold is separated into two or more sections, the part can be removed. However, certain geometrical

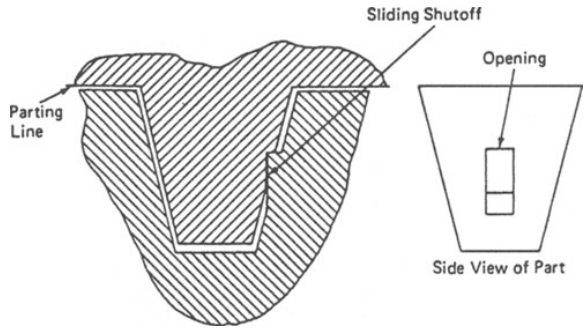


Fig. 4-117 Eliminating undercut side cores by using a sliding shutoff system.

considerations must be met. First, the part must have no undercut sections that will lock if it is pulled from the mold.

If such a shape is essential to function, a much more complicated mold is required in which a portion of the mold is retracted to permit the undercut to be removed. This complicates the molding procedure and the mold, and may result in higher costs as well as a poorer-quality part. The part will usually have some surfaces that are nearly parallel and perpendicular to the opening surface of the mold (referred to as the parting line), and pulling the part against these surfaces could result in sticking and drag that would make removal difficult and damage the product.

The product designer should restrict the number of undercuts to a minimum and consider carefully whether any undercuts in the design will present major problems in mold design. Moldings made from flexible plastic with small undercuts often allow forced mold release; that is, during mold opening the molding distorts sufficiently, because of its flexibility, to jump free of the undercut. This method is not recommended without experience. In such cases, a certain degree of permanent deformation may have to be accepted. Generously rounded corners are a must if this method of mold release is used.

For rigid plastics and large undercuts, use must be made of movable or rotating side cores, which obviously influence mold construction. Screw threads are an example of an undercut frequently met. To eliminate undercuts, consider tapering a wall so that a sliding shutoff can be used (Fig. 4-117).

Molded parts with undercuts (i.e., articles that cannot be released in the direction of the mold opening) require molds with more than one parting line. For such articles, various methods have been developed that may be operated manually, mechanically, hydraulically, pneumatically, or electromechanically:

1. Molds with side cores (Figs. 4-118 to 4-120 and 4-4)

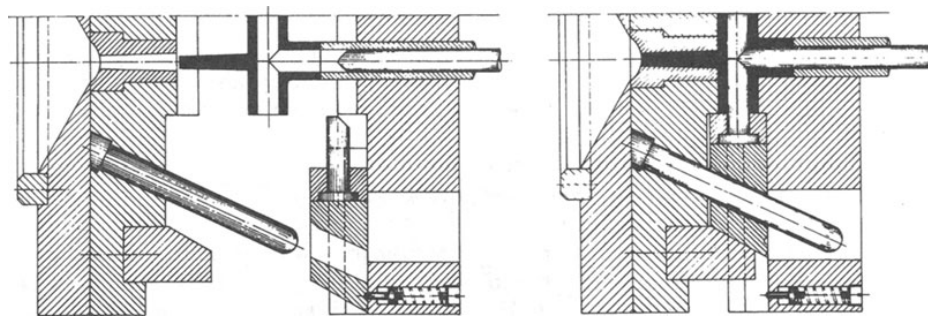


Fig. 4-118 Schematic of mold with mechanically actuated side cores.

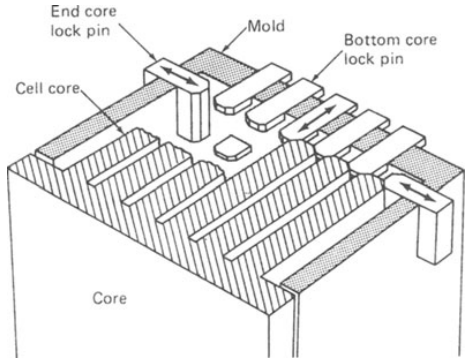


Fig. 4-119 Schematic of mold with side-core actions (battery case molded of PP).

2. Molds with wedges (Figs. 4-4 and 4-120)
3. Molds with rotating cores (Figs. 4-121 to 4-125)
4. Molds with loose cores or inserts

The choice of a method, or of a combination of these methods, is governed by not only the shape of the article and the properties of the polymer (flexibility, rigidity, shrinkage, etc.), but also the standards of quality to be met by the article (35). For articles with an external screw thread, for instance, either method 1 or 3 can be used. However, if method 1 is used, the mold parting line shows, which may be undesirable for aesthetic or design reasons. The method used should depend on the use of the article.

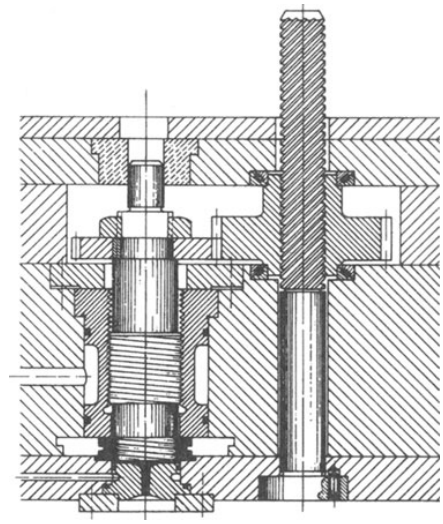


Fig. 4-121 Mold with rotating core that operates during mold opening and closing. The drive gear rotates via the worm shaft, which in turn transmits rotation to the geared core. The core unscrews the threaded molded product.

Mold Shrinkages and Tolerances

By shrinkage or tolerance is meant the dimensions to which a cavity and core should be fabricated in order to produce a product of desired shape and size (see Chap. 5, section on Molding Tolerances; Chap. 15, section on Micro Injection Molding). The usual way to decide on the amount of shrinkage is to consult data supplied by the material

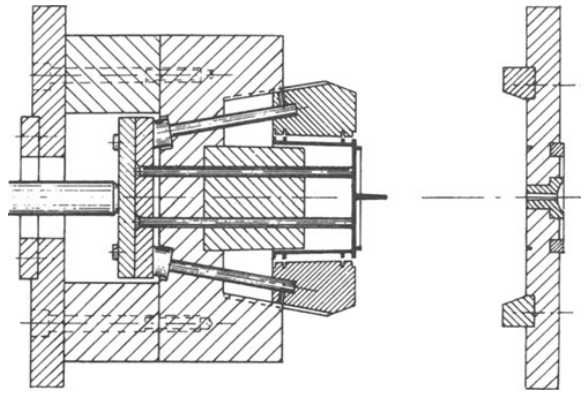


Fig. 4-120 Mold with wedge side-core action.

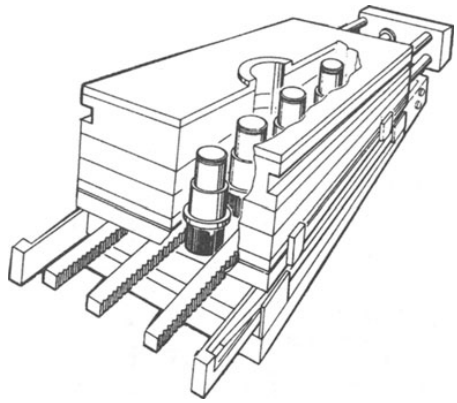


Fig. 4-122 Cores are positioned in rows as seen in this cutaway view of a closed mold frame. Each core resides within a gear that, when engaged by one of the parallel racks, causes the core to rotate and unscrew the molded caps.

manufacturer. The supplier's information is obtained from a test bar molded according to an ASTM standard (Chap. 12). The test bar is molded at a specific pressure, mold temperature, melt temperature, and cure time. The thickness of the test bar is normally $\frac{1}{8}$ in. (0.32 cm). However, molded parts are very rarely produced under conditions and sizes that are the same as or even similar to those used for test bars.

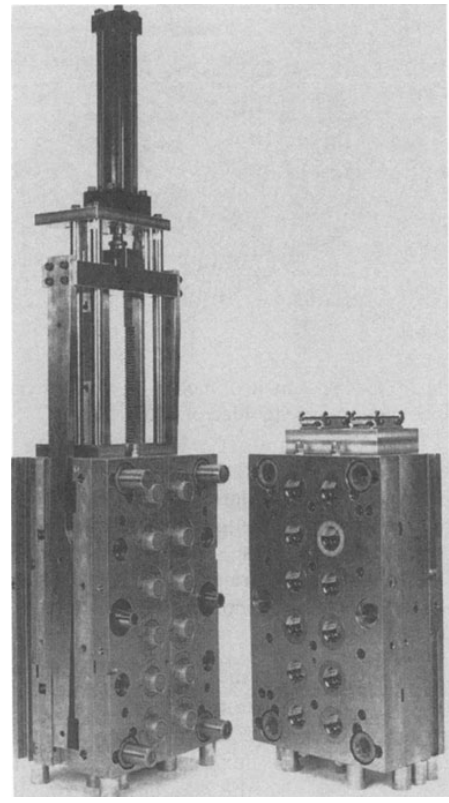


Fig. 4-124 Rack-and-pinion unscrewing mold for tamper-evident closures incorporates 12 rotating-core cavities.

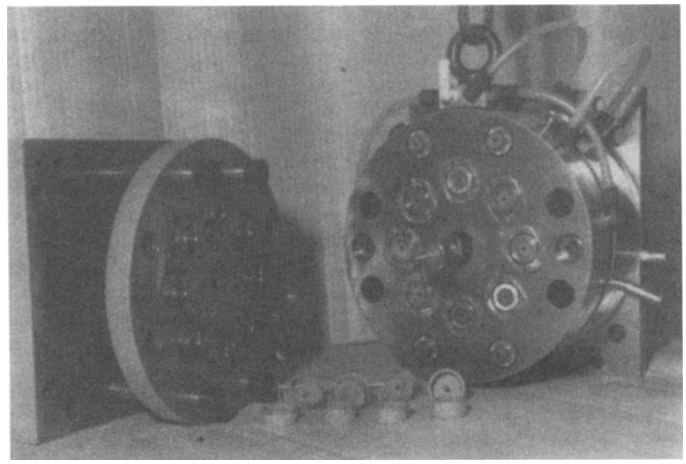


Fig. 4-123 Cap-unscrewing mold that has a rotating action for removal of molded caps. When the mold opens, the pin extending to the right side of the mold follows a guide plate and provides the rotating action for ejection.

DNA Genotek Inc. Exhibit 2029 Page 133
Spectrum Solutions LLC v. DNA Genotek Inc. IPR2023-01424

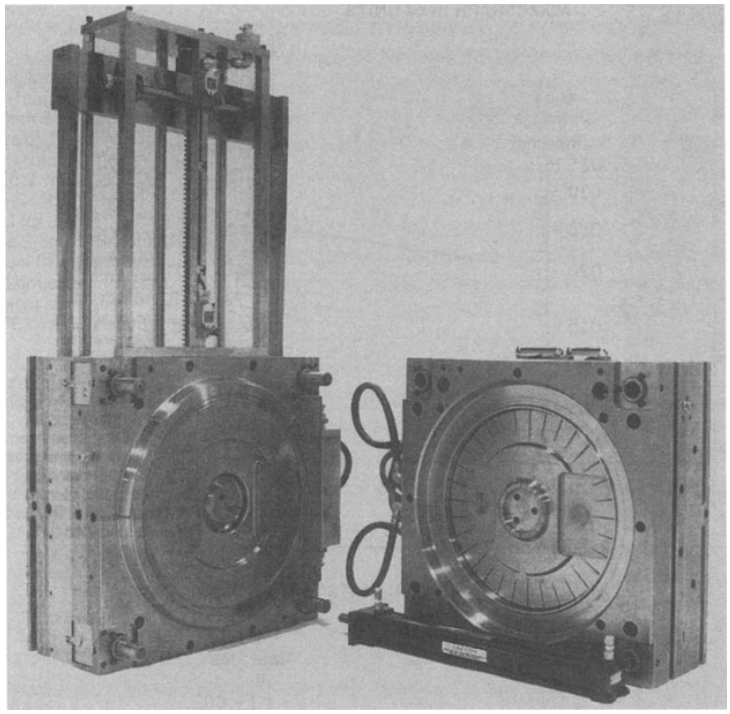


Fig. 4-125 Industrial container lid mold. Rack-and-pinion mechanism rotates a core section for the threaded inspection hole in the lid.

For precision parts with close tolerance dimensions, shrinkage information from test bars as furnished by material suppliers can be inadequate, but useful as a guide. We must become familiar with the factors that influence shrinkage so that we may arrive at more exact dimensions for a specific part. (See Chap. 8 on process effects.) According to compiled data, shrinkage is a function of mold temperature, part thickness, injection pressure, and melt temperature.

Shrinkage is influenced by cavity pressure to a very large degree. Depending on the pressure in the cavity alone, the shrinkage may vary as much as 100%.

Part thickness will cause a change in shrinkage. A thicker piece [$\frac{1}{8}$ in. (0.32 cm) or more] will have a shrinkage value on the high side of the data, whereas a thin one [$\frac{1}{20}$ in. (0.13 cm) or less] will have a lower shrinkage value.

The mold and melt temperature also influence shrinkage. A cooler mold will result in less shrinkage, whereas a hotter melt will cause more shrinkage, compared to the supplier's information.

The longer the time in the cavity, the closer the part comes to mold dimensions, which means a lessening of shrinkage.

Openings in the part will cause a variation in shrinkage from section to section because the cores making these openings act as temporary cooling blocks, which prevent a change in dimension while the part is solidifying. A relatively large gate size will permit higher cavity-pressure buildup, which brings about lower shrinkage.

The shrinkage problem can be categorized as follows:

1. Amorphous materials with a shrinkage of 0.008 in./in. or less have readily predictable shrinkage, which is not difficult to adjust with molding parameters such as cavity pressure and mold or melt temperature or, as a last resort, with the cycle.

DNA Genotek Inc. Exhibit 2029 Page 134
Spectrum Solutions LLC v. DNA Genotek Inc. IPR2023-01424

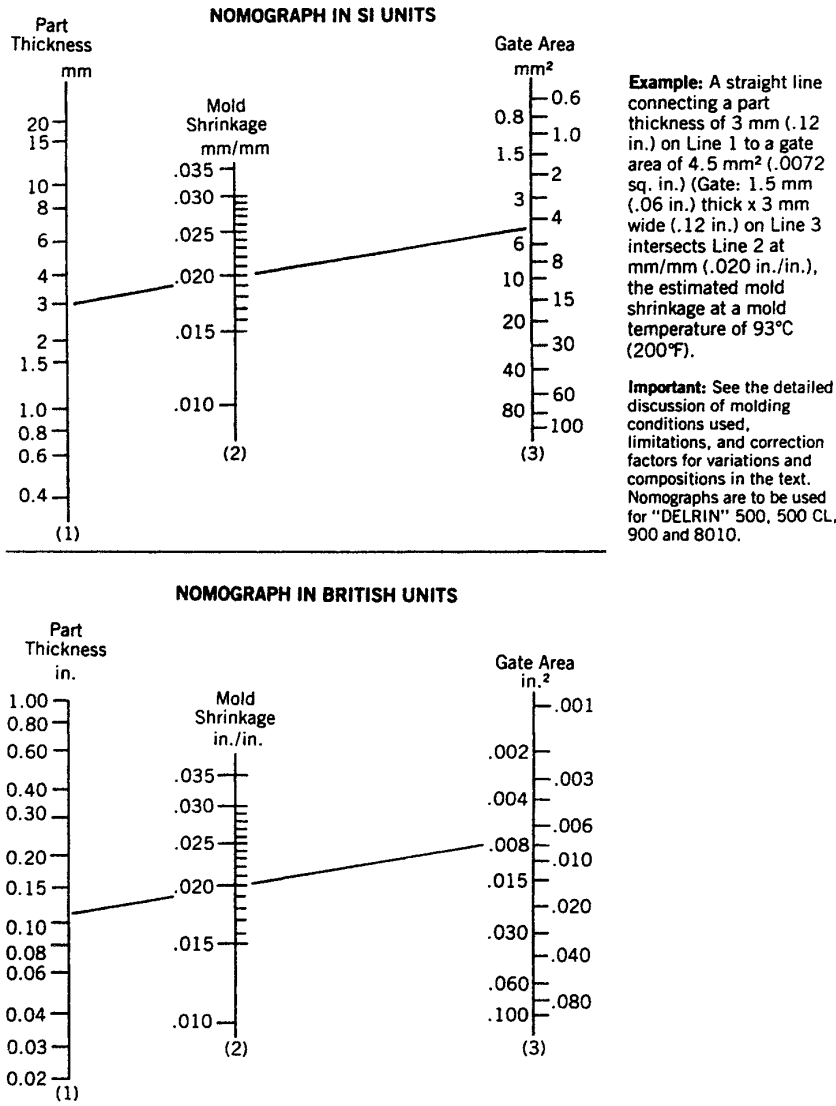


Fig. 4-126 Estimating mold shrinkage for DuPont Delrin acetal plastics.

2. Parts made of crystalline materials with high shrinkage (above 0.010 in./in.), but which are symmetrical and suitable for center gating, will also have a readily predictable shrinkage, adjustable with molding parameters.

3. Parts made of materials with a high shrinkage rate that are symmetrical but cannot be center-gated may approximate a center-gate condition if multiple gating close to the center (three, four, or six gates) is possible. In this case, the prediction of shrinkage is somewhat more difficult but still presents a chance of success.

4. The major problem exists with materials that have a high shrinkage rate (about 0.015 to 0.035 in./in.). In most of these cases, the material suppliers either show nomographs (see Fig. 4-126 and Chap. 6) in which all contributing factors are drawn and coordinated

to supply reasonably close shrinkage information, or they point to examples with actual shrinkage information and molding parameters that can be used for comparative interpolation. With most high-shrinkage crystalline materials (e.g., nylon, polyethylene, and acetal), when the material is edge- or side-gated, a larger shrinkage occurs in the direction of flow and a smaller one perpendicular to it. (See Chap. 6 regarding amorphous and crystalline plastics.)

If, upon review of the shrinkage information, there is still doubt about whether the precision dimensions will be attained, then there is one way left for establishing accurate shrinkage data: prototyping. In this method, a single cavity is built, and the critical dimensions are so calculated that they will allow for correction after testing, by providing for metal removal (machining). The test sample should be run for at least half an hour and under the same conditions as a production run. Only the last half-dozen pieces from the run should be used for dimensioning.

It is best to make the measurements after a 24-h period at room temperature. However, with crystalline thermoplastics, such as acetal, nylon, thermoplastic polyester, polyethylene, and polypropylene, the shrinkage may continue for days, weeks, months, or even a year. The shrinkage noted 1 h after molding may be only 75 to 95% of the total.

The reason for postmolding shrinkage is that there is a molecular rearrangement and stress relaxation going on until equilibrium is attained, at which point shrinkage stops; both the molecular rearrangement and stresses are brought about by molding conditions. The conditions that are most favorable for reaching the ultimate shrinkage in the shortest time are high mold temperature and a low rate of freezing. Each material has its own rate of postmolding shrinkage as a function of time. Curves showing the rate of shrinkage as a function of time for different mold temperatures and part thicknesses are available from material suppliers.

The upper range of mold temperature (shown in material-processing data sheets, Chap. 6) is most conducive to stopping shrinkage in the shortest time after part removal from the mold. Slow heat removal from the part is also desirable. Thin parts, which by their nature have a relatively fast heat-removal rate, consequently have a relatively long shrinkage stabilization time. It is to be emphasized that the problem of postmolding shrinkage exists principally with crystalline or semicrystalline thermoplastics.

The configuration of a product's end use and assembly will determine to what degree postmolding shrinkage will be a factor and what steps have to be taken to overcome a potential problem.

On some critical parts, an annealing (stress-relieving) operation may be necessary to offset possible dimensional changes. Remember that each crystalline material has a different postmolding shrinkage stabilization time.

Shrinkage vs. Cycle Time

Reducing the cycle time on injection machines means not only greater part output, but also, in some cases, less total capital investment in tooling and manpower for a given job. One important way to achieve such savings is to optimize cycles by properly sizing a tool based on realistic shrinkage rates for the material being run.

Although this sounds like a simple concept, its implementation often requires considerable experience. It also demands a willingness to take more time before production startup to employ the traditional engineering method of "trial and error." It is not for molders who are working within tight capital constraints. Generally, its payoff is only worthwhile for those high-volume jobs where the fast cycling characteristics of crystalline engineering thermoplastics is encountered.

The basic idea is simple: Before going into actual production, you try to cycle a part as fast as possible commensurate with quality requirements, measure the material shrinkage at the optimum cycle, and size your tool accordingly. There are several ways to approach this situation.

Almost all plastic material data sheets list shrinkage values, calculated for an average

DNA Genotek Inc. Exhibit 2029 Page 136
Spectrum Solutions LLC v. DNA Genotek Inc. IPR2023-01424

conservative molding cycle. They are only of marginal practical use, since it is usually possible to speed up cycles. And because shrinkage goes up as cycle time goes down—owing to faster cooling—every cycle time below a certain "conservative" value quoted on the data sheet yields a specific shrinkage value. There are other factors: mold design, method of cooling, wall thickness, and more. However, it is wrong just to depend on published data for sizing your tool, unless you plan to run the part at the exact same cycle suggested by the resin supplier for a specific wall thickness.

For all trials, a starting point must be established. A rough rule of thumb is that for crystalline materials such as nylon and acetal, shrinkage will run at about 0.015 and 0.020 in./in., respectively, for an average $\frac{1}{8}$ in. of wall thickness, at a conservative cycle time of 45 sec. Except for wall thickness, overall part size is not much of a limiting factor. With this in mind, Du Pont's Technical Services Laboratory, in Wilmington, Delaware, developed a nomograph, including a number of variables in the molding process, that will provide some refined information on shrinkage for a select number of materials (Fig. 4-126). Such a nomograph—along with experience with certain parts and resins—can be helpful in arriving at a starting point in sizing the tool.

From then on, it is mostly trial and error, but careful analysis of part design and optimum cycles also is very important. There are no tables you can use to determine meaningful "typical" shrinkage values at very fast cycles. You must determine them yourself at different cycle times. Following are two actual cases, illustrating several possible approaches.

Acetal part In this case, we were dealing with a flat part with precise hole-to-hole openings and a slide area that must be held open and flat. The resin was acetal with a shrinkage range from 0.02 to 0.035 in./in. The objective for the molder was to size the cavity so that a minimum cycle could be attained at proper dimensions. Typical cycles for this material might range from 30 to 60 sec. However, in order for the part to run at optimum cycle, mold shrinkage had to be chosen carefully for the part to stay in dimensions at very fast cycles.

When the cycle is longer than average for this material, shrinkage on such a part as described above will normally be about 0.02 in./in. at a cycle of about 60 sec, allowing for a long time of cooling. If the part is to be molded as quickly as possible, shrinkage increases to about 0.035 in./in. for a 30-sec cycle. (See Table 4-13).

In this case, the molder was in a position to use the fastest cycles. His prototype tool was sophisticated enough to run at various cycles, including the 30-sec rate. The molder then cut his production tool to 0.035 in./in.—after having established this figure as accurate for a 30-sec cycle in trials—and the parts were all on-size and acceptable. However, that also meant that the cycle could not be slowed down, or parts would tend to become oversized.

Table 4-13 Acetal flat part with precise hole-to-hole openings

	Long Cycle: 60 sec, Low Shrinkage	Short Cycle: 30 sec, Low Shrinkage	Short Cycle: 30 sec, High Shrinkage
Core pin spacing, in.	7.142	7.142	7.253
Required hole spacing, in.	7.0 ± 0.015	7.0 ± 0.015	7.0 ± 0.015
Corresponding hole spacing, in.	7.0	6.892	7.0
Designed mold shrinkage, in./in.	0.020	0.020	0.035
Actual mold shrinkage, in./in.	0.020	0.035	0.035

Resin is a grade of Delrin acetal with a 0.15-in. nominal wall thickness.

DNA Genotek Inc. Exhibit 2029 Page 137
Spectrum Solutions LLC v. DNA Genotek Inc. IPR2023-01424

Table 4-14 Nylon[a] doughnut-shaped product

	Long Cycle: 45 sec, Low Shrinkage	Short Cycle: 25 sec,[b] Low Shrinkage	Short Cycle: 25 sec, High Shrinkage
Core diameter, in.	1.629	1.629	1.652
Cavity diameter, in.	2.036	2.036	2.065
Corresponding part size:			
ID, in.	1.6	1.577	1.6
OD, in.	2.0	1.971	2.0
Designed mold shrinkage, in./in.	0.018	0.018	0.032
Actual mold shrinkage, in./in.	0.018	0.032	0.032

[a] Resin is a toughened nylon, Zytel ST 801 with a 0.2-in. nominal wall thickness.
[b] Part is undersized.

Nylon part In this case, the part was circular with a 2-in. (5.08-cm) outer diameter and inner diameter of 1.6 in. (4.06 cm), leaving a wall thickness of 0.2 in. (0.51 cm). Let us assume that the tolerance requirements on the inner diameter are such that the part must be held within 0.006 in. (0.015 cm). Therefore, the inner diameter dimension is 1.6 ± 0.003 in. (4.06 ± 0.0076 cm). Tolerance on the 2-in. outer diameter dimension is somewhat wider: ± 0.005 in. (0.013 cm).

The resin used is a toughened nylon. The range of mold shrinkages applicable to this resin and part could be as low as 0.018 in./in. or as high as 0.032 in./in., depending on the molding cycle.

In order to achieve the lower shrinkage, 0.018 in./in., the cycle would have to be fairly long, about 45 sec. On the other hand, at a 25-sec cycle—well within the capability of this particular resin—shrinkage could be as high as 0.032 in./in. (see Table 4-14). Therefore, the molder must choose the cycle time first, then determine the shrinkage at that cycle.

For instance, should the molder choose to run the fastest cycle, 25 sec, she would have to cut her mold for a 0.032-in./in. shrinkage. This, of course, assumes that there are no artificial limitations to achieving the fast cycle: lack of screw plasticating capacity, mechanical function of the mold, slow machine function, or improper cooling. The other consideration is the inability to predict the shrinkage value accurately at the given cycle time. Ideally, the molder would have experience with similar type parts; thus, she would be able to predict shrinkage at a specific cycle without trial and error methods.

Unfortunately, in most cases this is impossible. The part may be new, and the molder may have to go through a series of tests to establish the optimum cycle and corresponding shrinkage in order to size her tool properly. There are several approaches she can use.

With this particular part, the molder solved the shrinkage problem at the prototype tool stage. A prototype cavity, properly cored, that runs automatically at various cycles is the most reliable means of arriving at the minimum cycle. Such a cavity provides highly reliable data, provided the cooling for the prototype equals that on the production tool.

Cooling of the prototype tool is important. The more information you need from your prototype tool, the more sophisticated it must be—a simple Kirksite tool will not do for any extensive evaluation. A prototype tool probably should be made of P-20 steel, the same steel used for many production tools. The only difference is that the prototype tool is unhardened—vs. 55 Rockwell C for production tools—permitting machining ease. Although all this evaluation work is costly, a good prototype can save money in the long run; apart from optimizing cycles, with a tool that closely approximates the production tool, problems such as molded-in stresses due to improper gating or unequal

DNA Genotek Inc. Exhibit 2029 Page 138
Spectrum Solutions LLC v. DNA Genotek Inc. IPR2023-01424

rate of cooling can be detected and solved before full production startup. If, by proper choice of the shrinkage, you are able to cut 20 sec off a 60-sec cycle, this means a 33% productivity improvement.

A second, similar approach involves constructing a so-called lead cavity for the production mold. (For instance, on a four-cavity tool you would cut only one cavity for trial runs.) The part is then molded at various cycles to determine the shrinkage. This information can be translated into sizing the remaining cavities. The advantage is that this lead cavity will be designed to operate as a production tool, and all information learned is fully applicable to the other cavities.

Often, insufficient time is allowed for constructing the production tool, so the use of a lead cavity is impossible. In that case, serious thought should be given to the advisability of slowing down the initial development work. The time and money invested to develop cycle–shrinkage information can be amply rewarded, in higher productivity for the life of the part.

A third possible approach is to size the core piece for maximum shrinkage and size the cavity section of the production mold for minimum shrinkage. Referring to the circular part described above, we assume that the inner diameter will shrink 0.032 in./in. Applying this figure to the 1.6-in. (4.06-cm) inner diameter indicates that the core size should be cut to 1.652 in. (4.196 cm). If the cavity is sized to minimum shrinkage, 0.018 in./in., its size should be 2.036 in. (5.17 cm). With this technique, it is possible to remove steel from the core and cavity after the mold has been fully tested and the actual rate of shrinkage at a given cycle has been established. Although this is a somewhat time-consuming approach, it is by far less costly and time-consuming than other methods described previously.

A fourth approach would be to choose a reasonable shrinkage value, based on available data similar to Fig. 4-126, and cut both core and cavity to that value and cycle the part accordingly. For example, we might choose a shrinkage of 0.026 in./in. and pick a cycle time also in the midrange. If there are surprises, as can happen with any new part, the cycle and shrinkage can be adjusted in molding to achieve the required part size. In this particular case, good cycle information is not available from the prototype tool. It is possible to produce parts to the required size at the given cycle, but there is no way of knowing whether this cycle represents the full potential of the resin and tool. It would make more sense economically to cut both cores and cavity to achieve the final part dimension at a fast cycle rather than at a middle value. The cavity work is rather easy to accomplish, since it only entails increasing its size by cutting steel. The cores would have to be remachined from scratch because they must be larger.

Shrinkage can vary with cycle time, so that final part dimensions depend on the precise cycle time for a mold. Therefore, before you size your tool, determine how fast you can effectively run the part. Find out the shrinkage at that cycle. Based on these findings, your final tool dimensions will then be keyed to production at optimum cycle time.

In some cases, of course, a too rapid cycle can interfere with some secondary operation, such as an operator assembly procedure required at every ejection. This situation would require that the molder go the other way, resizing the cavity to slow down the cycle.

As a molder, you have to be aware of potential quality defects caused by fast cycles. For example, very fast cycles may prevent thick sections from fully packing out, resulting in voids. Before you decide to push a cycle time to its limit, you must decide if part quality will suffer. Good candidates for minimum cycle times are usually thin-walled parts and those with uniform wall thickness.

Ejection of Molded Products

Adherence of parts on the injection half requires the placing of cores and other retaining devices on the moving half of the mold so that there is no chance of parts hanging up in the cavity. Even a slight tendency to stick in any portion of the cavity will cause warpage, stresses, and dimensional distortion of parts. Such a tendency may indicate a need for additional taper, polish lines in the direction

DNA Genotek Inc. Exhibit 2029 Page 139
Spectrum Solutions LLC v. DNA Genotek Inc. IPR2023-01424

of withdrawal, or manipulation of the mold temperature.

In dimensioning the cavity and core, close attention is to be given to ensuring the unstressed retention of the part on the ejection side. This is normally accomplished by the plastic shrinking tightly over the cores and adhering to them. In such cases, it is desirable to have a rougher surface on the core than in the cavities. In some configurations, it becomes desirable to provide narrow undercuts 0.002 to 0.005 in. (0.0051 to 0.0127 cm) deep in the area of the ejection pins.

All surfaces in line with the mold opening direction, such as sidewalls, etc., must have a certain draft to facilitate ejection of the molding from the mold. Insufficient draft can cause deformation or damage. The draft required for mold release is primarily dependent on the depth of the cavity: the deeper the cavity, the more draft necessary. In the determination of the draft required, shrinkage (which differs for each plastic) must also be taken into account. If metal inserts are used, shrinkage can have an adverse effect on mold release, which can be prevented by using more draft.

Another factor affecting mold release is the rigidity of the molding; rigid moldings require less draft than more flexible ones. In general it is recommended that a minimum draft of one degree be used. For small moldings, a draft of one-half to one degree may be sufficient in some cases, whereas for large moldings drafts up to three degrees may be required.

The ejection of a molding is generally by ejector pins, which are commercially available in various designs and qualities, or by strips, bushings, plates, or rings (Fig. 4-127). The choice of ejector system is largely governed by article shape, and by the rigidity or flexibility of the plastic used. Whatever ejector system is chosen, ejection must never cause damage to, or permanent deformation of, the molding. The mold preferably should be fitted with ejectors at those spots around which the molding is expected to shrink (i.e., around cores or male plugs).

The force required to strip a molding off a male core may be determined approximately

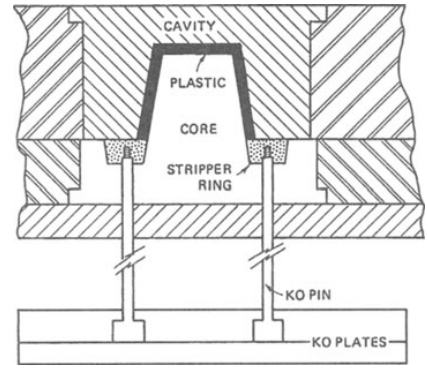


Fig. 4-127 Stripper-ring ejection.

from

$$p = \frac{S_t \times E \times A \times \mu}{d\left(\frac{d}{2t} - \frac{d}{4t} \times \gamma\right)}$$

where P = ejection force required (lbf or kgf)
E = elastic modulus (lbf/sq in. or kgf/sq cm)
A = total area of contact between molding and mold faces in line of draw (sq in. or sq cm)
μ = coefficient of friction between plastic and steel
d = diameter of circle circumference equal to perimeter of molding surrounding male core (in. or cm)
t = thickness of molding (in. or cm)
γ = Poisson's ratio of the plastic
S_t = (thermal contraction of plastic across diameter d) = (coefficient of thermal expansion) × (temperature difference between softening point and ejection temperature) × d (in. or cm)

At high mold temperatures, allowance must be made for thermal expansion of the mold platens. These platens will expand more than those of the ejector mechanism. It is recommended that the ejectors be provided with a cylindrical head, and they should be mounted with some clearance to allow the correction of possible variations in center distances during machine operation.

DNA Genotek Inc. Exhibit 2029 Page 140
Spectrum Solutions LLC v. DNA Genotek Inc. IPR2023-01424

Ejection of articles with large cylindrical or flat surfaces may sometimes be hampered, as such surfaces tend to create a vacuum between the articles and cavity wall. In such cases, mold release may be improved and the vacuum broken by an air ejection system operated by air valves or channels, in the mold and valve stem, operating independently or in conjunction with ejector pins.

Related to an ejector is a sprue puller, which is a device used to pull (draw) a molded sprue out of the sprue bushing. It is generally a straight round pin with the end machined in the form of an undercut. There are different methods in general use for providing an anchor to pull the sprue. The sprue lock pin is located where the small depression at the mold entrance meets the runners. It is fastened to the knockout or ejector mechanism and runs through the movable part of the mold in direct line with the mold entrance.

Mold Release Agents

A mold release agent, also called a parting agent, is a substance, such as silicone, put upon an interior mold cavity surface and/or added to a molding compound, to facilitate removal of the molded product from the mold. Note that using certain agents, including silicone, can cause bonding problems if parts are to be decorated in a secondary operation, or may interfere with electrical circuits.

Mold Materials of Construction

Choices of material range from computer-generated plastic tooling to specialty alloys or even pure carbide. Everyone from purchasing agents to shop personnel must consider the ramifications of tool materials. They include factors such as construction, hardness, corrosion resistance, wear resistance, product design, productivity, output rate, tool maintenance cost, and life. One may consider the softest tool that will do the job because it is usually the least expensive to build, but that requires special/careful handling and still have limited life. There are also basic questions that have to be answered in advance of any specifications of tool materials.

The chief materials of construction are various grades of steels. Others include beryllium copper alloy, brass, aluminum, kirksite, sintered metal, steel-filled epoxy plastic, and flexible plastic. As a guide to life expectancy consider P-20 steel for one million parts, QC-7 aluminum for 250,000, sintered metal for 100,000, and filled epoxy plastic for 50 to 200 (1, 140, 143, 179, 209, 295, 525, 526, 508, 588).

Steels

Commonly used is P 20 steel, a high grade of forged tool steel relatively free of defects and prehardened. It can be textured or polished to almost any desired finish and is a tough mold material. H-13 is usually the next most popular mold steel used. Stainless steel, such as 420 SS, is the best choice for optimum polishing and corrosion resistance. Other steels and materials are used to meet specific requirements, such as copper alloys for fast cooling, aluminum for extended mold life and low cost, etc. (404).

Different characteristics and performances identify steels. As an example, higher hardness of steel improves wear, dent, and scratch resistance and polishability, but lowers machinability and weldability. High sulfur content degrades the stainless qualities and polishability of the steel. Hardness, as a measure of the internal state of stress of the steel, has an adverse effect on weldability, fracture toughness, and dimensional stability.

The steel used in the manufacture of a mold base varies, depending on the requirements of the application. (As an example, polyvinyl chloride requires stainless steel to eliminate corrosion.) The structural sections of the mold base are usually made from medium carbon (SAE 1030) or AISI-4130-type steels. Among the steels selected for cavity and core plates are P-20- and H-13-type steels, as well as stainless steel (T-420) (Table 4-16).

The available spectrum of modern tool steel offers properties in numerous combinations and to widely differing degrees. Fortunately, the needs of the vast majority of

DNA Genotek Inc. Exhibit 2029 Page 141
Spectrum Solutions LLC v. DNA Genotek Inc. IPR2023-01424

Table 4-15 SPI Moldmakers Division quotation guide

THE MOLDMAKERS DIVISION
THE SOCIETY OF THE PLASTICS INDUSTRY, INC.
3150 Des Plaines Avenue (River Road), Des Plaines, Ill. 60018, Telephone: 312/297-6150

TO _____ FROM _____ QUOTE NO. _____
_____ _____ DATE _____
_____ _____ DELIVERY REQ _____

Gentlemen:
Please submit your quotation for a mold as per following specifications and drawings:

COMPANY NAME _____

Name of Part/s
1. _____ B/P No. _____ Rev. No. _____ No. Cav. _____
2. _____ B/P No. _____ Rev. No. _____ No. Cav. _____
3. _____ B/P No. _____ Rev. No. _____ No. Cav. _____

No. of Cavities:	Design Charges:	Price:	Delivery:

Type of Mold: ☐ Injection ☐ Compression ☐ Transfer ☐ Other (specify) _____

Mold Construction
☐ Standard
☐ 3 Plate
☐ Stripper
☐ Hot Runner
☐ Insulated Runner
☐ Other (Specify) _____

Mold Base Steel
☐ #1
☐ #2
☐ #3

Special Features
☐ Leader Pins & Bushings in K.O. Bar
☐ Spring Loaded K.O. Bar
☐ Inserts Molded in Place
☐ Spring Loaded Plate
☐ Knockout Bar on Stationary Side
☐ Accelerated K.O.
☐ Positive K.O. Return
☐ Hyd. Operated K.O. Bar
☐ Parting Line Locks
☐ Double Ejection
☐ Other (Specify) _____

Material

Cavities	Cores
☐ Tool Steel	☐
☐ Beryl. Copper	☐
☐ Steel Sinkings	☐
☐ Other (Specify) _____	

Press
Clamp Tons _____
Make/Model _____

Hardness

Cavities	Cores
☐ Hardened	☐
☐ Pre-Hard	☐
☐ Other (Specify) _____	

Finish

Cavities	Cores
☐ SPE/SPI	☐
☐ Mach. Finish	☐
☐ Chrome Plate	☐
☐ Texture	☐
☐ Other (Specify) _____	

Cooling

Cavities	Core
☐ Inserts	☐
☐ Retainer Plates	☐
☐ Other Plates	☐
☐ Bubblers	☐
☐ Other (Specify) _____	

Ejection

Cavities	Cores
☐ K.O. Pins	☐
☐ Blade K.O.	☐
☐ Sleeve	☐
☐ Stripper	☐
☐ Air	☐
☐ Special Lifts	☐
☐ Unscrewing (Auto)	☐
☐ Removable Inserts (Hand)	☐
☐ Other (Specify) _____	

Side Action

Cavities	Cores
☐ Angle Pin	☐
☐ Hydraulic Cyl.	☐
☐ Air Cyl.	☐
☐ Positive Lock	☐
☐ Cam	☐
☐ K.O. Activated Spring Ld.	☐
☐ Other (Specify) _____	☐

Type of Gate
☐ Edge
☐ Center Sprue
☐ Sub-Gate
☐ Pin Point
☐ Other (Specify) _____

Design by: ☐ Moldmaker ☐ Customer
Type of Design: ☐ Detailed Design ☐ Layout Only
Limit Switches: ☐ Supplied by _____ ☐ Mounted by Moldmaker
Engraving: ☐ Yes ☐ No
Approximate Mold Size: _____
Heaters Supplied By: ☐ Moldmaker ☐ Customer
Duplicating Casts By: ☐Moldmaker ☐Customer
Mold Function Try-Out By: ☐ Moldmaker ☐ Customer
Tooling Model/s or Master/s By: ☐ Moldmaker ☐ Customer
Try-Out Material Supplied By: ☐ Moldmaker ☐ Customer

Terms subject to Purchase Agreement. This quotation holds for 30 days.

Special Instructions: _____

The prices quoted are on the basis of piece part print, models or designs submitted or supplied. Should there be any change in the final design, prices are subject to change.

By _____ Title _____

Distribution: Use of this 3 part form is recommended as follows: 1) White and yellow - sent with request to quote. Pink - maintained in active file. 2) White original - returned with quotation. Yellow - retained in Moldmaker's active file.

tool-steel users can be satisfied with a relatively small number of these steels, the most widely used of which have been given the identifying numbers of the American Iron and Steel Institute (AISI).

With reengineered molds and components, manufacturers can provide performance and capabilities based on mold design requirements. It is important in the process of mold purchasing to develop professional forms that detail special mold design features, as well as steel types, heat treatments, and surface finish requirements. Sample forms have been developed (Table 4-15). The

DNA Genotek Inc. Exhibit 2029 Page 142
Spectrum Solutions LLC v. DNA Genotek Inc. IPR2023-01424

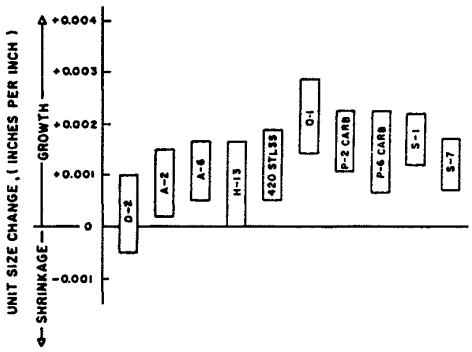


Fig. 4-128 Anticipated range of changes in heat treatment of various mold steels.

information presented in Fig. 4-128 and Tables 4-16 through 4-21 characterizes materials that are useful in work with preengineered molds and components, as well as your own mold.

Tool steels (or mold steels) may be defined as highly alloyed steels. The chemistry and method of manufacture determine the use of the final product. As high-performance alloys such as cobalt, vanadium, and chromium become more difficult and more expensive to obtain, it is safe to say that the quality of the product and better usage will be the key to the future of mold steels.

Proper materials selection and proper combination of alloys in varying percentages are required for finished tools. Characteristics of machinability, wear, shock, and antigalling properties, resistance to corrosion, and, of course, hardenability are directly attributable to alloy type and content.

Combining the chemistry of alloys with the best melting, rolling, and annealing techniques allows steel mills to consistently produce fine-quality tool steels. Tool steels are electric-furnace-melted and, when beneficial, vacuum-degassed on pouring. Detailed annealing procedures produce structures that also enhance fine quality. This quality is of vital importance to the moldmaker, who must produce molds of the highest caliber repetitively.

The following is an indication of where most common mold steels and beryllium copper find application in injection molds, along with comments on each material's particular strengths or weaknesses.

Type 4130, 4140 The type most commonly used in a prehardened state at a hardness of

Table 4-16 Examples of steel applications in molds

Type of Steel	Typical Uses in Injection Molds
4130/4140	General mold baseplates
P-20	High-grade mold baseplates, hot-runner manifolds, large cavities and cores, gibs, slides, interlocks
4414 SS, 420 SS (prehardened)	Best-grade mold baseplates (no plating required), large cores, cavities, and inserts
P5, P6	Hobbed cavities
01	Gibs, slides, wear plates
06	Gibs, slides, wear plates, stripper rings
H-13	Cavities, cores, inserts, ejector pins, and sleeves (nitrided)
S7	Cavities, cores, inserts, stripper rings
A2	Small inserts in high-wear areas
A6	Cavities, cores, inserts for high-wear areas
A10	Excellent for high-wear areas, gibs, interlocks, wedge locks, wedges
D2	Cavities, cores, runner, and gate inserts for abrasive plastics
420 SS	Best all-around cavity, core, and insert steel; best polishability
440C SS	Small- to medium-size cavities, cores, inserts, stripper rings
250, 350	Highest toughness for cavities, cores, small unsupported inserts
455M SS	High toughness for cavities, cores, inserts
M2	Small core pins, ejector pins, ejector blades (up to $\frac{5}{8}$-in. diameter)
ASP 30	Best high-strength steel for tail, unsupported cores, and core pins

DNA Genotek Inc. Exhibit 2029 Page 143
Spectrum Solutions LLC v. DNA Genotek Inc. IPR2023-01424

Table 4-17 Identification of steels by AISI-SAE designations

Steels	AISI-SAE Designations[a]	Amount of Alloy
Carbon	10XX	Plain with 1.00% maximum Mn
	11XX	Resulfurized
	12XX	Resulfurized and rephosphorized
	15XX	Plan with 1.00 to 1.65% Mn
Manganese	13XX	1.65% Mn
Nickel	23XX	3.50% Ni
	25XX	5.00% Ni
Nickel–chromium	31XX	1.25% Ni; 0.65 to 0.80% Cr
	32XX	1.75 Ni; 1.07% Cr
	33XX	3.50% Ni; 1.50 to 1.57% Cr
	34XX	3.00% Ni; 0.77% Cr
Molybdenum	40XX	0.20 or 0.25% Mo
	44XX	0.40 or 0.52% Mo
Chromium–molybdenum	41XX	0.50, 0.80, or 0.95% Cr; 0.12, 0.20, 0.25, or 0.30% Mo
Nickel–chromium–molybdenum	43XX	1.82% Ni; 0.50 or 0.80% Cr; 0.25% Mo
	47XX	1.05% Ni; 0.45% Cr; 0.20 or 0.35% Mo
	81XX	0.30% Ni; 0.40% Cr; 0.12% Mo
	86XX	0.55% Ni; 0.50% Cr; 0.20% Mo
	87XX	0.55% Ni; 0.50% Cr; 0.25% Mo
	88XX	0.55% Ni; 0.50% Cr; 0.35% Mo
	93XX	3.25% Ni; 1.20% Cr; 0.12% Mo
	94XX	0.45% Ni; 0.40% Cr; 0.12% Mo
	97XX	0.55% Ni; 0.20% Cr; 0.20% Mo
	98XX	1.00% Ni; 0.80% Cr; 0.25% Mo
Nickel–molybdenum	46XX	0.85 or 1.82% Ni; 0.20 or 0.25% Mo
	48XX	3.50% Ni; 0.25% Mo
Chromium	50XX	0.27, 0.40, 0.50, or 0.65% Cr
	51XX	0.80, 0.87, 0.92, 0.95, 1.00, or 1.05% Cr
Chromium with 1.00% C min	50XXX	0.50% Cr
	51XXX	1.02% Cr
	52XXX	1.45% Cr
Chromium–vanadium	61XX	0.60, 0.80, or 0.95% Cr; 0.10 or 0.15% V minimum
Tungsten–chromium	72XX	0.75% Cr; 1.75% W
Silicon–manganese	92XX	0.00 or 0.65% Cr; 0.65, 0.82, or 0.85% Mn; 1.40 or 2.00% Si

[a] The first two digits denote the type of steel, and the second two digits the carbon percentage.

30 to 36 Rc for load- or pressure-bearing mold base plates, such as cavity and core retainer plates, or extralarge cavities and cores, which have no special surface-quality requirements.

Type P-20 This type is the same as 4130/4140; however, its cleaner manufacturing requirement results in a more homogeneous microstructure and, thus, good polishability. It is used for large cavities and cores requiring good polish, and for hot-runner manifolds.

Type 414 SS / 420 SS prehardened This type is most commonly used at a hardness of 30 to 35 Rc; it is excellent for large cavities and cores that require a good polished finish and corrosion resistance. It is also very good for cavity and core retainer mold base plates, providing toughness and corrosion resistance without a need for plating.

DNA Genotek Inc. Exhibit 2029 Page 144
Spectrum Solutions LLC v. DNA Genotek Inc. IPR2023-01424

Table 4-18 Materials used in molds, arranged in decreasing order of surface hardness

Suitable for			Material Class	Suitable for		
Thermoplastics glass-filled	Low-pressure thermosets SMC BMC	High-pressure thermosets Phenolics Ureas Diallylls Melamines Alkyds	Carbides Steel, nitriding Steel, carburizing Steel, water-hardening Steel, oil-hardening Steel, air-hardening Nickel, cobalt alloy Steel, prehardened 44 Rc Beryllium, copper	Thermoplastics unfilled		
			Steel, prehardened 28 Rc Aluminum bronze Steel, low alloy & carbon		Blow molds	
	Prototype injection molds TP resins	Structural foams	Kirksite (zinc alloy) Aluminum, alloy			Vacuum-forming sheets TP resins
		Casting of liquid resins	Brass Sprayed metal Epoxy, metal-filled Silicone, rubber			

P-5 and P-6 Carburizing steel available in an annealed condition. It is easy to hob and/or machine for making cavities, and can be carburized to a depth of 60 mils and case hardness of 58 to 61 RD. Its relatively soft core (15 to 30 Rc) lowers the overall compressive strength, which is a key quality in modern mold-steel requirements. In the long run, it is often more economical to pay for the higher mold manufacturing cost of ED-Med tool steel cavities of the through-hardened type, rather than using the hobbing process, because of the much longer life expectancy.

01 Oil-hardened Available in an annealed condition, this material is capable of attaining a maximum of 62 Rc hardness. It is excellent for gibs, slides, wear plates, and the like, but not recommended for cavity or core components or mold base plates.

06 Oil-hardened This type has the same applications as 01, but provides better machinability and especially good wear characteristics in applications with metal-to-metal contact, because of the presence of free graphite in its microstructure.

H-13 Air-hardened One of the most useful steels for moldmaking, this material provides good all-around steel qualities for cavities and cores, as well as inserts.

S-7 Air-hardened This is the same as H-13, but provides the often required higher hardness of 54 to 56 Rc. Extreme care is required in the heat-treating process; to prevent cracking in the quench, a double draw is highly recommended. It is very important also that a hardness of 55 Rc be achieved accurately, because of this steel's very sharp breakoff point in impact strength or toughness (highest at 55 Rc, lowest at 57 to 58 Rc).

Types A2, A6, A10 air-hardened Medium-alloy tool steels available in an annealed condition. A2 is the most abrasion-resistant steel of this group under molding conditions, because of its higher chrome content. A10 has remarkable wear and nonseizing qualities in metal-to-metal contact applications, because of its free graphite content. All three are easy to machine and very high in compressive strength. Welding, however, can create cracking problems.

D2 Air-hardened This material is in a class by itself with respect to excellent abrasion resistance and is recommended for

DNA Genotek Inc. Exhibit 2029 Page 145
Spectrum Solutions LLC v. DNA Genotek Inc. IPR2023-01424

Table 4-19 General characteristics of typical mold steels

AISI Type	Trade[a] Designation	General Characteristics	Property Rankings[b] Toughness	Dimensional Stability in Heat Treatment	Machinability (Annealed)	Polishability (Heat-treated)	Typical Applications
P-20	CSM-2	Medium carbon (0.30%) and chrome (1.65%). Available prehardened (300 Bhn), or annealed (200 Bhn).	10	7	9 (prehardened)	8 (prehardened)	Excellent balance of properties for injection and compression molds of any size.
H-13	NuDie V	Hot-work die steel; 5% chrome. May be hardened to about 50 Rc.	9	8	9	9	Higher hardness than P-20; good toughness and polishability. Used for abrasion resistance in RP molds and high-finish injection molds.
A-2	Air Kool	Cold-work die steel, high carbon (1.0%) 5% chrome. May be hardened to about 60 Rc.	8	9	8	7	High hardness for abrasion-resistant, long-wearing compression and injection molds. Limited to small sizes.
D-2	Airdi 150	Cold-work die steel; high carbon (1.55%) 11.5% chrome. May be hardened to about 60 Rc.	7	9	5	6	Highest abrasion resistance. Difficult to machine. Susceptible to stress cracking. Small molds only.
414	CSM 414	Stainless steel; 12% chrome, 2% Ni, 1% Mn, low carbon (0.03%). Available prehardened (300 Bhn)	10	10	9	9	"stainless version" of P-20; similar properties and uses.
420	CSM 420	Stainless steel; 13% chrome, 0.80% Mn, medium carbon (0.30%). May be hardened to about 50 Rc).	9	10	8	10	"Stainless version" of H-13; similar properties and uses. Very stable in heat treatment, takes high polish.
4145	Holder block	Medium carbon (0.50%) and chrome (0.65%). Available prehardened.	10	10(prehardened)	10(prehardened)	6 (prehardened) 7 (fully hardened)	Low-cost steel, for mold bases and large molds. Not suited to high-quality finish.

[a] Crucible steel designations.
[b] On scale of 1 to 10 (10 = best).

DNA Genotek Inc. Exhibit 2029 Page 146
Spectrum Solutions LLC v. DNA Genotek Inc. IPR2023-01424

Table 4-20 Important properties of the major mold steels and beryllium copper

Type	AISI Designation	Recommended Hardness, Rockwell C	Point Ratings, 1 to 10 (10 is Highest)											
			Wear Resistance	Toughness	Compressive Stress	Corro-Hot Hardness	sion Resistance	Thermal Conductivity	Hobbability	Machinability	Polishability	Heat-treatability	Weldability	Nitriding ability
Prehardened	4130/41040	30–36	2	8	4	3	1	5	1	5	5	10	4	4
	P-20	30–36	2	9	4	3	2	5	1	5	8	10	4	5
Prehardened stainless	414 SS	30–35	3	9	4	3	7	2	1	4	9	10	4	6
	420 SS	30–35	3	9	4	3	6	2	1	4	9	10	4	7
Carburizing	P-5	59–61	8	6	6	5	2	3	9	10	7	6	9	8
	P-6	58–60	8	7	6	5	3	3	8	10	7	6	8	8
Oil-hardening	01	58–62	8	3	9	5	1	5	5	8	8	7	2	3
	06	58–60	8	4	8	5	1	5	7	10	5	6	2	3
Air-hardening	H-13	50–52	6	7	7	8	3	4	6	9	8	8	5	10
	S7	54–56	7	5	8	8	3	4	6	9	8	8	3	8
	A2	56–60	9	3	9	7	3	4	4	8	7	9	2	8
	A5	56–60	8	4	8	7	2	5	5	10	7	7	4	7
	A10	58–60	9	5	9	7	2	5	5	8	6	7	2	NA
	D2	56–58	10	3	8	8	4	2	4	4	6	9	1	10
Stainless	420 SS	50–52	6	6	6	8	7	2	4	7	10	8	6	8
	440C SS	56–58	8	3	8	7	8	2	3	6	9	7	4	NA
Maraging	250	50–52	5	10	6	7	4	3	4	4	7	9	5	9
	350	52–54	6	10	7	7	4	3	4	4	7	9	5	9
Maraging stainless	455M	46–48	5	10	5	7	10	2	3	4	8	9	5	NA
High-speed	M2	60–62	10	2	10	10	3	3	2	4	6	8	2	10
	ASP 30	64–66	10	4	10	10	4	3	1	4	7	8	2	NA
Beryllium copper	Be Cu	28–32	1–2	1	2	4	6	10	10	10	8–9	7	7	NA

NA = not available.

DNA Genotek Inc. Exhibit 2029 Page 147
Spectrum Solutions LLC v. DNA Genotek Inc. IPR2023-01424

Table 4-21 Effects of alloying materials

Element	Symbol	Description
Aluminum	Al	Combines with nickel and titanium to form an intermetallic compound, which precipitates on aging and provides strength and hardness. Also used as a deoxidizer and to produce fine grain size.
Carbon	C	Very influential in controlling hardness, depth of hardness, and strength.
Chromium	Cr	A carbide-forming element that contributes strongly to hardenability and abrasion and wear resistance. Additional amounts of chromium, greater than are needed for carbide formation, remain in solution and enhance corrosion resistance.
Cobalt	Co	An element added to the maraging steels to improve strength.
Manganese	Mn	Combines with free sulfur to form discrete sulfide inclusions and improve hot workability. It is also a deoxidizing agent. In larger quantities, it increases hardenability by decreasing the required quenching rate. It is the principal element used to obtain quenching by air cooling, which minimizes distortion.
Molybdenum	Mo	Promotes hardenability in mold steels. The elevated tempering requirement increases the steel's strength at higher operating temperatures and provides more complete relief of residual stresses for greater dimensional stability.
Nickel	Ni	Usually added to improve hardenability of low-alloy steels. In maraging steels, nickel combines with aluminum and titanium to form an intermetallic compound that increases hardness and strength on aging. Larger amounts of nickel also assist in corrosion resistance.
Silicon	Si	Principal function is as a deoxidizing agent during melting. In higher quantities, it retards tempering, thus allowing higher tempering and operating temperatures (hot hardness).
Titanium	Ti	Found in maraging steels, where it acts as a potent strengthener by combining with nickel and/or aluminum to form an intermetallic compound, which precipitates on aging.
Tungsten	W	Increases hardness, strength, and toughness.
Vanadium	V	A strong carbide-forming element, which is usually added to control grain size and increase wear resistance.

severe molding conditions, such as when glass or mineral fillers are used. It is not recommended for welding and is somewhat sensitive to cracking, owing to its low toughness.

Types 420 and 440C stainless These are good choices for corrosion resistance when corrosive plastics are used, or moisture or humidity could affect cavity surface finish or cooling-channel corrosion. Type 440C is somewhat better in wear resistance and compressive strength, owing to its higher hardness, whereas 420 SS represents the true mold cavity steel with good to very good all-around qualities and exceptionally high and consistent polishability, provided that it is manufactured by vacuum degassing and/or electroslag remelting. Its low thermal conductivity compared with other mold steels is only a minor factor in the first few days or weeks of processing a new mold. As soon as corrosion inside cooling channels takes hold, the thermal conductivity of other mold steels, with respect to cooling-channel effectiveness, will be worse than that of stainless steels.

Maraging types 250, 350, 440M These are excellent mold cavity and insert steels. They are by far the best performers when toughness is the priority, as in cases of very thin cross sections or small, fragile, and unsupported cavity or core inserts. Their resistance to cracking could, in the long life of the mold, be a crucial factor in mold-repair expenses,

DNA Genotek Inc. Exhibit 2029 Page 148
Spectrum Solutions LLC v. DNA Genotek Inc. IPR2023-01424

offsetting the much higher initial price of these steels (five to ten times that of other tool steels). Dimensional stability and simplicity of heat-treating these steels are valuable considerations for the moldmaker.

M2 and ASP 30 high-speed steels Probably the most useful of all the many high-speed steels for moldmaking, M2 is by far the most useful steel for good-quality, long-lasting round core pins or blade ejectors, and is also readily available. ASP 30 is an advanced-generation steel manufactured by Uddeholm, using a new powder-metallurgy process. Its extremely high density gives it remarkable rigidity, which can be very important in resisting the deflection of tall, unsupported cores.

Soft tools In general soft tooling can be anything other than the usual steels used in production molds. It includes materials such as cast or machined aluminum grades, cast plastics (epoxy, silicone, etc.), cast rubbers, and cast zincs. Soft tools are the least expensive, and the most flexible in application. They are usually faster to fabricate, but have limited lives compared to steel molds. Today's choices range from computer-generated plastic molds to specialty alloys or even pure carbide. However, each of them has limitations in durability and capabilities.

These molds can last a fairly long time if they are properly prepared and maintained. Steel wear-resistant edge plates can be used to extend their life expectancy. Preventative maintenance, such as cleaning, is very important. To clean, use a mold cleaner designed to loosen normal parting line and vent residue. The cleaning fluid should be compatible with the tooling material (197).

Heat Treating

As progress has been made in the quality of tool steels and mold construction, so have advances in heat treating. Knowledge has expanded, and the development of new equipment such as vacuum furnaces, fluid-bed furnaces, and finer tempering facilities has made the heat-treating operation much more of a science than ever before.

Many times this procedure appears to be taken for granted, yet it is one of the most important. In investigative analysis, 70% of all tool failures are related to heat treating, and it is not always the fault of the hardener. Of greater concern is the fact that one-half of these failures are due to poor surface conditions, the bane of all molders.

Requirements to be Met by Mold Steel

Machinability Molds are usually formed by the machining of steel blocks. In view of the high cost, the steel must possess good machinability, which depends on the composition and structure of the steel as supplied.

Ability to harden In general, the hardening of small molds or mold components does not present great problems. However, hardening of large and complicated molds may cause deformation, dimensional variances, or even cracks, if in the selection of the tool steel insufficient allowance was made for the hardening treatment, machining techniques, and dimensions of the mold components (size and shape). The material must be capable of being hardened without any risk. Depending on the hardening process, the following steel grades may be used: oil-hardened steel, air-hardened steel, pre-heat-treated steel, case-hardened steel, and nitrided steel.

Ability to take a polish The surface finish of the molding is first and foremost governed by the mold cavity finish. A cavity polished to a mirror finish produces a glossy molding surface and assists polymer flow in the mold. Polishing ability depends on the hardness, purity, and structure of the tool steel used. High-carbide steel grades are hard to polish to a mirror finish and thus require additional labor.

Corrosion resistance If corrosive plastics are processed, proper corrosion resistance of the mold steel is a must. Even the slightest

DNA Genotek Inc. Exhibit 2029 Page 149
Spectrum Solutions LLC v. DNA Genotek Inc. IPR2023-01424

corrosion of the mold cavity will interfere with mold release and surface finish on molded product.

Aluminum

Compared to steel, aluminum is usually considered by molders to be too soft, with limited durability and shortcomings relative to compressive strength, reparability, and surface finish. However, current grades have overcome these concerns. In many cases good design practices can compensate for these limitations. With the automobile industry driving everything as far as large molds are concerned, production expectations have come down dramatically. It is common to see aluminum plate molds that have run anywhere from 500,000 to one million parts. Because the thermal conductivity of Al is higher than that of steel, the cycle time is reduced. From a handling and operating approach, the lighter-weight Al provides about a 3 : 1 weight advantage. Other advantages include lower-cost and much faster machining than tool steel. (1, 259, 263, 320).

Beryllium–Copper

BeCu is used in molds to provide relatively fast heat transfer. There are two basic families of BeCu alloy: those with high heat conductivity and those with high strength. The heat conductivity of the former is about 10 times greater than those of stainless steels and tool steels. It is double that of aluminum alloys such as alloy 7075, and higher than that of others.

BeCu alloys have higher hardness and strength than aluminum. When heat-treated, they are the strongest of all copper-based alloys. They are not usually recommended for high-production molds, because of their relatively low wear resistance, toughness, and compressive strength, compared with tool steel. They do, however, have a special place in moldmaking when economy in cavity manufacturing and injection molding cycle time (the latter minimized by BeCu's high thermal conductivity) are of the utmost importance. However, one must take into consideration that, over the lifetime of a mold, periodic cavity replacement costs can become a great disadvantage.

Kirksite

In processes where pressures are low and short runs are anticipated (usually in the thousands), kirksite molds can be used. Because the material pours so well, it is generally cast, and type-A kirksite is usually used. Since the pouring temperatures are low [800°F (427°C) for kirksite as compared to 3,000°F (1,650°C) for steel and 2,000°F (1,093°C) for beryllium–copper castings], it is possible to cast copper tubing cooling lines directly into place in kirksite.

More important, the low casting temperatures (and retention of fluidity for a relatively long period of time) enable kirksite to pick up fine detail from the pattern over a very large casting area. This means that kirksite molds will reproduce pattern detail in the molded parts (it falls somewhere between aluminum and beryllium–copper in this regard) and thus has found application when fine patterns such as wood grains are required (e.g., furniture parts). Shrinkage is about 0.008 in./in.

Kirksite is lower in cost than most other metals and machines well. It is nonmagnetic and therefore may need clamping for grinding. It has a tendency to load grinding wheels badly. Kirksite is also heavier than aluminum and only slightly lower in weight than steel or beryllium copper. It is not as strong as either of those other metals and therefore will require heavier wall sections, making it more difficult to handle. Cycles with kirksite molds are usually shorter than with steel, but longer than with aluminum molds.

Brass

This alloy of copper and zinc is used in the manufacture of molds, dies, instruments, etc. One of its desired and excellent properties is good heat transfer.

DNA Genotek Inc. Exhibit 2029 Page 150
Spectrum Solutions LLC v. DNA Genotek Inc. IPR2023-01424

Etching Cavity Surfaces

Etching, also called photoetching, chemical blanking, or chemical machining, is a controlled chemical process that depends on the action of an acid or alkali, depending on the type of material (plastic, metal, etc.), that uniformly attacks all exposed areas of the product. A mask or protective coating is used on those surfaces that are not to be etched.

As an example, a wide variety of molded parts can be produced with a pattern or textured surface reproduced from a mold photoetching. Attractive patterns range from leather and wood grain to line patterns with varying directions and depths. Basically, the required pattern is transferred to the mold by a photographic process. The pattern is then etched to the required depth by the application of an appropriate acid, under closely controlled conditions. It can be performed either on complete tools/molds or on specific areas.

Factors that influence results on the different tool steels (H13, 420, etc.) are: (1) grades, annealing, and hardening; (2) flame hardening, welding, EDM, etc.; (3) grain flow direction of the tool steel; (4) variations in tool steel and cleanliness; and (5) material size. If nitriding is to be used, it must be done after etching. Flame hardening prior to etching should be avoided, since the pattern will be etched differently in the flame-hardened zone. Welded steel can usually be etched if the same steel is used in the weld. Poor etching occurs on surfaces marred by residual traces of spark machining, grind, or polish. Steels with a clean microstructure and low sulfur content give the most accurate and consistent pattern.

Machining Safety

Any machining process that generates airborne, respirable particles is cause of concern, regardless of the material being machined. OSHA publishes guidelines for the amount of exposure to respirable particles workers should not exceed. The list includes stainless steel, H13, P20, and many alloying elements (including chromium, vanadium, nickel, copper, molybdenum, and beryllium). To be hazardous, these particles must be smaller than 10 μm, and thus are not visible to the naked eye. The large, easily visible particles or chips generated in most machining operations do not represent an inhalation hazard (Chap. 2, section on Safety; Chapter 10, section on Machining).

Moldmaker Directory

In this age of specialization, the purchasing community has found it increasingly difficult to locate the right source for the job. As an assist, the Moldmakers Section of the SPI provides industry an updated directory of its members and their special capabilities. The SPI moldmaker members are in constant contact with the plastics industry and its ever-changing technology. The directory lists moldmakers as contract or custom services and in turn by type of process mold, such as injection molding and blow molding.

Mold Material Selection Software

Different software programs are available from different sources, usually material producers. An example is the PLA-Ace software package from Daido Steel Co., Tokyo, Japan. It provides the basic information that encompass selections that include a mold base, cavity, and core pin(s). In addition it provides different inputs such as type of plastic being processed, plastic content (glass fiber, flame-retardant, etc.), cavity texture or finish, comparative wear and hardness capabilities, minimum mold thickness, corner radius, existence of ribs, product category (electronic, auto, etc.), product type (panel, TV chassis, etc.), and cause of mold defects due to improper material selection both for Daido's tool steels and for those of other suppliers. Such software can help processors and tool shops with limited experience select the optimum material (Chap. 9, section on Computer Software). It also serves as a useful tutorial tool for trainees (175).

DNA Genotek Inc. Exhibit 2029 Page 151
Spectrum Solutions LLC v. DNA Genotek Inc. IPR2023-01424

Fabrication of Components

Mold cavity and core inserts are fabricated by a wide variety of methods (293, 520). To date, however, conventional machining of the cavity and core inserts accounts for the most widely employed method of fabrication. The term machining is used here to denote milling, duplicating, drilling, boring, turning, grinding, and cutting.

The first step after selection of the cavity and core material is to cut the raw material to the approximate size. This is normally accomplished with either vertical or horizontal power sawing equipment. The next operation is to square, true, and size the inserts. Normally, this stage of fabrication is carried out in the material's soft or annealed stage if hardened steels are being utilized for the final product. Round inserts generally are trued and sized on turning equipment, which includes lathes and cylindrical grinders. Square and rectangular blocks are milled when the hardness of the raw material allows them to be. Final sizing of prehardened or hardened blocks is completed on surface grinders.

In this preliminary stage of fabrication, the heels required for retention in through pockets are established, or screw holes are installed for inserts that will fit into blind pockets. Although in these preliminary stages it appears that not much progress is being made on the cavities and cores, squaring is one of the most important steps in mold manufacture. Available from mold suppliers are tool steel inserts that have been sized from 0.005 to 0.015 oversize for square or rectangular blocks, with round inserts furnished with heels. Considerable savings of in-house labor are possible by utilizing these inserts, which are generally available in P-20, H-13, and 420 stainless steel.

One of the next operations to be carried out on the inserts is installation of the mold temperature control circuits. Square and rectangular blocks normally have water, steam, or oil channels drilled directly into them, with other lines connecting to form internal loops. Frequently, internal water channels are blocked with threadless brass pressure plugs to direct the flow of the temperature control fluid. Generally, only one inlet and one outlet are used per insert, with the remaining channels blocked using brass or steel pipe plugs. Conventional drilling equipment is normally utilized for this operation, with gun drilling used for deep holes where accuracy of location is required. Great care must be taken in this operation, as nothing dampens the enthusiasm of the moldmaker more than hitting a water line with a screw hole.

As modern molds require greater sophistication in temperature control to meet the challenges presented by today's design engineers, programs have been developed and are commercially available from suppliers to accurately predict the amount and placement of temperature-control channels. Generally, the core will be required to remove approximately 67% of the heat generated in the injection molding process. This requirement presents one of the greatest challenges to the moldmaker, as less space usually is available in the core than in the cavity.

A generation ago, the standard of the industry was the reliable Bridgeport, with the skill of the journeyman moldmaker coaxing accuracy from the equipment. Today, the standard is the NC, CNC, or DNC milling, drilling, duplicating, EDM, and grinding machines. Often, tapes are generated by the mold designer, a practice made possible by the wide acceptance of CAD/CAM equipment in even the smallest of operations. The addition of CAD/CAM equipment has made the skilled craftsman more productive and more valuable.

Accuracy of modern equipment or skill of the journeyman moldmaker is required for the machining operations that now will be incorporated in the cavity and core inserts. Depending on subsequent machining operations, the knockout (ejector) pin holes may be located and established at this time. Perhaps one of the greatest values in moldmaking today is the off-the-shelf availability of high-precision ejector pins. Nitrided hot-work ejector pins, constructed from superior-quality thermal stock H-13 steel, are available in the shoulder or straight type with an outside diameter hardness in the 65 to 74 Rc range. Ejector pins, both imperial and

DNA Genotek Inc. Exhibit 2029 Page 152
Spectrum Solutions LLC v. DNA Genotek Inc. IPR2023-01424

metric, starting at $\frac{3}{64}$ (0.046) in. and ranging to 1 in., are available in every popular fraction, letter, and millimeter size required by the moldmaker. Should requirements dictate nonstandard sizes, such pins also can be built by the component supplier as a special. For purposes of future replacement, the use of standard, off-the-shelf components is highly recommended and frequently demanded by most tool engineers.

Hobbing

Cavities are formed by methods other than machining. After machining, hobbing ranks as the next most common method. Hobbing consists of forcing a hardened negative pattern into the cavity stock under extremely high pressure. The obtainable cavity depth is limited to a certain fraction of the hob diameter. Hobs frequently are constructed from steels such as S-1, S-4, A-2, A-6, and D-2. The master hob must contain the polish and precise detail desired in the finished cavity, as the process faithfully reproduces the master hob's characteristics in the cavities.

The next operation after the hobbing itself is to cut the outside of the cavity blank to the desired size, install any required water passages and gate details, harden, and then final-polish. Molds that contain high cavitation are the most viable candidates. The closure industry takes great advantage of this process because it uses many such molds.

Cast Cavities

Cast cavities have certain advantages not obtainable with either machining or hobbing. The advantage of cast cavities is that large amounts of excess stock do not have to be removed by machining. A variety of cavity materials are suitable for casting, including steel, beryllium copper, kirksite, aluminum, and others. In the casting process, a pattern must be constructed that not only incorporates the features of the desired cavity configuration, but also the shrinkage of the casting material, as well as that of the plastic which is to be molded. As it is almost always less expensive to machine a pattern than the final cavity, fabrication economics favor this process in larger cavities. The prototyping of large automotive parts is frequently done in cast kirksite molds. Because the melting point of kirksite is relatively low, patterns can be built from wood or plaster. Another advantage is that in some materials, water lines can be cast internally. As with many specialty processes, the casting of cavities is best performed by companies specializing in this process. Steels that are cast include AISI 1020, 1040, 4130, 4340, and 8630, as well as S-7 and stainless steel.

Electroforming

Electroforming is a process in which metal is deposited on a master in a plating bath. Many companies engaged in electroforming use proprietary processes, which are closely guarded. In one method of forming cavities, the master is constructed of plastic and coated with silver to provide a conductive coating. The coated master then is placed in a plating tank and nickel or nickel-cobalt is deposited to the desired thickness, which can approach $\frac{1}{4}$ in. (0.64 cm). With this method, a hardness of up to 46 Rc is obtainable. The nickel shell then is backed up with copper to a thickness sufficient to allow for machining a flat surface, to enable the cavity to be mounted into a cavity pocket.

The electroforming process is used for the production of single or small numbers of cavities, in contrast with hobbing. Some deep cavities are formed by this process instead of swaging, including long slender components such as ballpoint pen barrels.

Electric-Discharge Machining

Electric-discharge machining (EDM) is another widely utilized method of producing cavity and core stock. Electrodes fabricated from materials that are electrically conductive are turned, milled, ground, and developed in a large variety of shapes, which

DNA Genotek Inc. Exhibit 2029 Page 153
Spectrum Solutions LLC v. DNA Genotek Inc. IPR2023-01424

duplicate the configuration of the stock to be removed. The electrode materials, selected for their ability to be economically fabricated while producing the desired wear characteristics, include graphite, copper, tungsten, copper–tungsten, and other electrically conductive materials.

The use of EDM in the production of molds for plastics is firmly established. Development of the process has produced significant refinements in operating technique, productivity, and accuracy, while widening the versatility of the process. Wire EDM has emerged as an efficient and economical alternative to conventional machining. Special forms of EDM can now be used to polish tool cavities, produce undercuts, and make conical holes from cylindrical electrodes.

EDM, also called spark erosion, is a method involving electrical discharges between a graphite or copper anode and a cathode of tool steel or other tooling material in a dielectric medium. The discharges are controlled in such a way that erosion of the tool or workpiece takes place. During the operation, the anode works itself down into the workpiece, which thus acquires the same contours as the former. The dielectric flushing liquid is ionized during the course of the discharges. The positively charged ions strike the cathode, whereupon the temperature in the outermost layer of the steel rises so high (18,000 to 90,000°F) as to cause the steel there to melt or vaporize, forming tiny drops of molten metal that are flushed out as *chippings* into the dielectric. The craters (and occasionally also chippings that have not separated completely) are easily recognized in a cross section of a machined surface.

Four main factors need to guide the choice of operating parameters for an EDM operation on tool steel:

- The stock-removal rate
- The resultant surface finish
- Electrode wear
- The effects on the tool steel

The influence of the EDM operation on the surface properties of the machined material can, in unfavorable circumstances, jeopardize the working performance of the tool. In such cases, it may be necessary to subordinate the first three factors, when you are choosing machining parameters, in order to optimize the fourth.

Tooling

Tool is a general term that includes molds, mandrel, jigs, fixtures, punch dies, etc. for shaping and fabricating parts. Special tools are used to maintain the tooling. As an example, brass tooling is used to clear or remove melted plastic during processing that may be trapped in the hopper throat when melt bridges, sticks to a screw, etc. The brass does not damage the steel, as steel or other metals would. Beryllium tools are sometimes used, but they are harder than brass.

Polishing

Molds usually require a high polish. Though the operation seemingly is gentle, polishing can damage the steel unless it is properly done (369). A common defect is *orange-peel*, a wavy effect that results when the metal is stretched beyond its yield point by overpolishing and takes a permanent set. Attempts to improve the situation by further vigorous polishing only make matters worse; eventually, the small particles will break away from the surface. The more complicated the mold, the greater the problem.

Hard carburized or nitrided surfaces are much less prone to the problem. Orange-peel results from exceeding the yield point of the steel. The harder the steel, the higher the yield point and therefore the less chance of orange-peel.

The surest way to avoid orange-peel is to polish the mold by hand. With powered polishing equipment, it is easier to exceed the yield point of the metal. If power polishing is done, use light passes to avoid overstressing.

Orange-peel surfaces usually can be salvaged by the following procedure: Remove the defective surface with a very fine-grit stone; stress-relieve the mold; restone; and diamond-polish. If orange-peel recurs after

DNA Genotek Inc. Exhibit 2029 Page 154
Spectrum Solutions LLC v. DNA Genotek Inc. IPR2023-01424

this treatment, increase the surface hardness by nitriding with a case depth of no more than 0.005 in. and repolish the surface.

A large part of mold cost (typically 5 to 30%) is polishing cost. An experienced polisher can polish from 2 to 5 sq in./h (12.9 to 32.3 sq cm/h). Certain shops can at least double this rate if they have the equipment.

Polishing is rarely done for appearance alone. It is done to either obtain a desired surface effect on the part, facilitate the ejection of the product from the mold, or prepare the mold for another operation such as etching or plating. If a part is to be plated, it is important to remember that plating does not hide any flaws—it accentuates them. Therefore, on critical plated-part jobs, the mold polish must be better than for nonplated parts.

Another purpose of polishing is to remove the weak top layer of metal. It may be weak from the stresses induced by machining or from the annealing effect of the heat generated in cutting. When it is not removed, this layer very often breaks down, showing a pitted surface that looks corroded.

The techniques used to get a good and fast polish are basically simple, but they must be followed carefully to avoid problems. The first rule is to make sure the part is as smooth as possible before polishing. If electrical-discharge machining is used, the final pass should be made with a new electrode at the lowest amperage. If the part is cast or hobbed, the master should have a finish with half the roughness of the desired mold finish.

When the mold is machined, the last cut should be made at twice the normal speed, the slowest automatic feed, and a depth of 0.001 in. (0.0025 cm). No lubricant should be used in this last machining, but the cutting tool should be freshly sharpened, and the edges honed after sharpening. The clearance angle of the tool should be from 6 to 9 deg, and if a milling cutter or reamer is used, it should have a minimum of four flutes. A steady stream of dry air must be aimed at the cutting tool to move the chips away from the cutting edge.

Polishing a mold begins when the designer puts the finishing information on the drawing. Such terms as "mirror finish" and "high polish" are ambiguous. The only meaningful way is to use an accepted standard to describe what has to be polished and to what level. It is also important that the designer specify a level of polish no higher than is actually needed for the job, because going from just one level of average roughness to another greatly increases the cost of the mold. Figure 4-129 illustrates the cost of polishing as a function of the roughness of the finish.

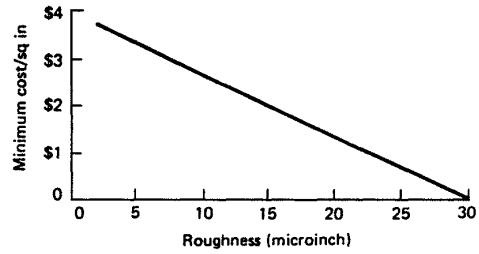


Fig. 4-129 Cost of polishing mold steels.

Roughness is given as the arithmetical average in microinches. A microinch is one-millionth of an inch (10^{-6}) and is the standard term used in the United States. Sometimes written as MU inch or μin., it is equal to 0.0254 micron. A micron, one-millionth of a meter (10^{-6} meter), is the standard term in countries that use the metric system. It is often written as micrometer or μm and is equal to 39.37 μin.

SPI Finish Numbers

There are several ways to specify a certain level of roughness. One common and important standard is the SPI–SPE Mold Finish Comparison Kit. It consists of six steel disks finished to various polish levels and covered with protective plastic caps processed in molds with those finishes. One disk has a roughness of 0 to 3 μin., another a roughness of 15 μin., and all the others are coarser. Because 15-μin. roughness is acceptable in fewer than 10% of all jobs, the result is that the disk with zero to 3-μin. roughness—the highest level of polish—must be selected in almost all other cases.

A no. 6 finish is achieved with 24-grit dry blast at 100 psi from a distance of 3 in. A

DNA Genotek Inc. Exhibit 2029 Page 155
Spectrum Solutions LLC v. DNA Genotek Inc. IPR2023-01424

Table 4-22 Diamond compound specifications

Diamond Compound	Finish	Particle Size (μin.)	Approximate Mesh	Color
1–8M	Super	0–2	14,000	Ivory
3–7M	Very high	2–4	8,000	Yellow
6–48M	Mirror	4–8	3,000	Orange
9–6M	High	8–12	1,800	Green
15–5M	Fine	12–22	1,200	Blue
30–4H	Lapped	22–36	600	Red

no. 5 finish is achieved with 240-grit dry blast at 100 psi from 5 in. A no. 4 finish is the achieved with a 280-grit abrasive stone. A no. 3 finish is achieved with a 320-grit abrasive cloth. A no. 2 finish is obtained when the final phase of polishing is completed using 1,200-grit diamond (up to 15-μm diamonds). The highest finish is the SPI no. 1 finish, resulting from 8,000-grit diamond (0- to 3-μm diamond range).

Another standard that has come into general use is the specifying of a finish as produced by a polishing compound containing diamond particles within a certain microinch range (Table 4-22). It is not a perfect system, but works in most cases.

A near-perfect system is the American Standard Association's standard ASA B 46.1. This corresponds to the Canadian standard CSA B 95 and British standard BS 1134. For a definition of the terms used in ASA B 46.1 and how to apply them to mold drawings, see Figs. 4-130 through 4-132. The use of this standard in specifying finishes leaves no room for disputes about what is called for, and since all quotes apply to the same standard, the polishing costs tend to be more uniform. The biggest drawback to its use is that many tool shops do not have the necessary test equipment.

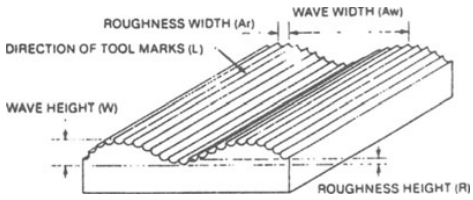


Fig. 4-130 Terms used in Standard ASA B 46.1 for measuring mold surface roughness.

The term "roughness cutoff width" refers to the distance the instrument checks to obtain the roughness values (which do not include the wave values). This distance should be long enough to measure all the irregularities except the waves. The standard values specified are 0.003, 0.010, 0.030, 0.100, 0.300, and 1 in. If no value is specified, then 0.030 in. is assumed. The metric equivalents are 0.075, 0.250, 0.750, and 7.5 mm.

To determine the wave width and height, both of which are caused by the cyclic instability of the machine doing the cutting, it is necessary to measure at least one wave width, which sometimes is as much as 1.5 in., or 40 mm. The wave height is measured as the maximum peak-to-valley distance. When only one value is specified for a particular characteristic, it should always be taken as the maximum.

Both the moldmaker and the designer should be aware of the types of finishes that can be obtained using various manufacturing methods. Table 4-23 contains a list of these methods.

Hand Benching

The first step in mold finishing normally involves the use of both hand tools and power-assisted grinders to prepare the surfaces to be polished. Depending on the last machining operation and roughness of the remaining stock, the moldmaker will select the method of preparing the surface in the quickest manner possible. Typical hand tools include files and diemaker rifflers. The files range from 20 to 80 teeth/in. and will assist in removing stock quickly and accurately. Rifflers are

DNA Genotek Inc. Exhibit 2029 Page 156
Spectrum Solutions LLC v. DNA Genotek Inc. IPR2023-01424

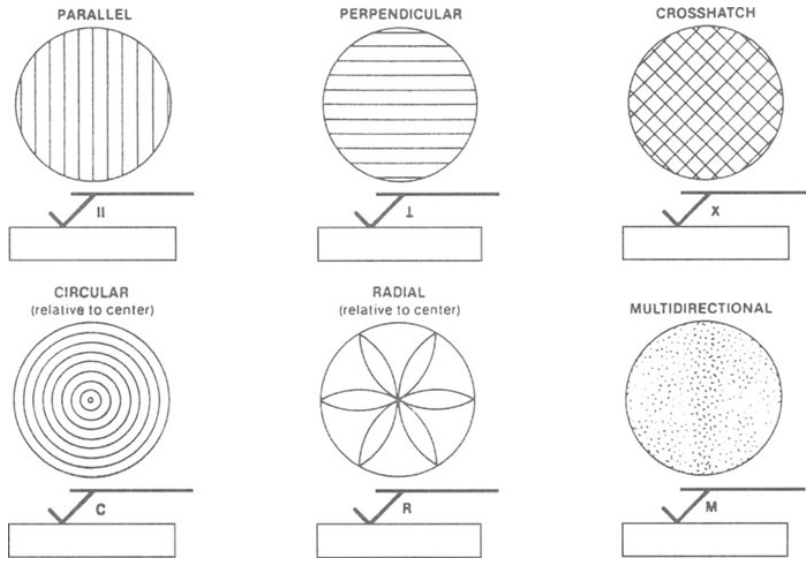


Fig. 4-131 Symbols use on mold drawings to indicate the lay pattern for polishing mold cavities, etc.

available in a wide range of shapes to fit into any conceivable contour found on a mold and have from 20 to 220 teeth/in.

Power-assisted tools include electric, pneumatic, and ultrasonic machines, which can be equipped with a wide variety of tool holders. Rotary pneumatic and electric grinders, with the tool mounted either directly or in a flexible shaft, are extremely popular with the moldmaker for rapidly removing large amounts of stock. With the grinding or cutting medium mounted in a 90° or straight tool, depending on the surface, the moldmaker will start with the coarser medium and work the surface progressively to the finer medium. The finishing process can best be described as one where each preceding operation imparts finer and finer scratches on the surface. The initial stages are perhaps the most important in the finishing state. Spending too little time in this phase normally will be detrimental to the final surface finish.

A variety of cutting tools, from carbide rotary cutters, abrasive drum, band, cartridge, tapered cone, and disk to flap wheels, are used to reduce the roughness of the surface in the quest for the desired mold surface finish. The use of abrasive stones, either worked by hand or in conjunction with reciprocating power tools, normally is the next phase in mold finishing. These stones are a combination of grit particles suspended in a bonding agent. For general-purpose stoning, silicon carbide is available in type-A stones for steel hardness under 40 Rc and type-B stones for higher hardness. Most stones are available in square, rectangular, triangular, and round shapes in grits of 150, 240, 320, 400, and 600. Other stones consist of aluminum oxide in type E for working EDM surfaces, type M that has oil impregnated for additional lubrication, and type F for added flexibility or reduced breakage of thin stones.

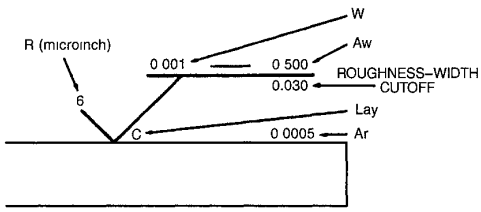


Fig. 4-132 Method of describing roughness at a given point on a mold surface. Symbols, see Fig. 4-130.

Direction of Benching

The next step normally consists in using abrasive sheets or disks to continue smoothing of the mold surface. All the finished work

Table 4-23 Surface roughness produced by various manufacturing methods

Process	Roughness Height (μin.)	Process	Roughness Height (μin.)
Flame cutting	250–2,000	Lapping	1–32
Snagging	125–2,000	Superfinishing	1–32
Sawing	63–2,000	Polishing	1–32
Drilling	63–500	Sand casting	250–2,000
EDM & CM	32–1,000	Ceramic casting	32–250
Milling	16–1,000	Investment casting	32–250
Turning	16–1,000	Pressure casting	32–125
Boring	16–1,000	Forging	63–500
Reaming	16–250	Diecasting	16–250
Tumbling	2–63	Injection molding	2–63
Grinding	2–125	Stamping	16–250
Honing	2–63		

is carried out in the direction of the scratches installed in the line of draw or ejection. Machine, file, stone, or abrasive marks installed perpendicular to the direction of draw are detrimental to removal of the part from either the cavity or core, and must be avoided. At this point, the desired mold surface finish may have been achieved, or the piece parts grit-blasted, and finishing is considered complete. These finishes may be acceptable for nonfunctional core surfaces in which ejection does not pose a problem.

Ultrasonic Tools

The latest aid for mold finishing has been the introduction of ultrasonic finishing and polishing systems. The ultrasonic controls deliver strokes that can be adjusted between 10 and 30 μm at speeds up to 22,000 cycles per minute. This action eliminates costly hand finishing and is extremely effective in polishing deep narrow ribs that have been EDM-installed. These devices, along with diamond files, ruby or abrasive stones, and wood laps, have made the work of the mold polisher much more pleasant.

Textured Cavities

Texturing can add another dimension to a molded part. For example, the term " camera case finish" has been used to describe a textured surface finish that is desirable in certain situations. A surface that is to be textured should be worked up to a 320-grit (SPI no. 3) finish. A technician then applies the selected pattern to the mold surface. In this process, acid will be applied to the area where mold material will be dissolved; therefore, wax or other resistant coating is applied to the surface that will remain on the cavity. Selective texturing on a cavity will require that acid-resistant material be applied to areas of no texturing to guard against attack by the acid. The depth of the texturing will be a function of the mold material, its chemical resistance, the type and concentration of the acid, and time. As with many other processes, the quality of the finished part depends on the person performing the task, who must frequently check the depth of the texture to ensure that the desired effect is achieved. When the operator is satisfied with the texture depth, the acid is neutralized, and the wax resist removed.

Patterns of Different Textures

Almost any pattern imaginable is possible. Most texture sources have a wide range of standard patterns, and molded plaques are available to show the final results. The designer must be careful to allow sufficient draft on the cavity walls for proper release of the plastic from the textured surfaces. First, the draft angles must be at least the minimum

DNA Genotek Inc. Exhibit 2029 Page 158
Spectrum Solutions LLC v. DNA Genotek Inc. IPR2023-01424

acceptable for the plastic that will be molded. Then, from 1 to $1\frac{1}{2}^{\circ}$ of draft per 0.001 in. (0.00025 cm) of texture depth must be allowed, to enable proper release of the textured plastic part. When special directional patterns, which may run perpendicular to the parting line, or low-shrinkage, less flexible materials are to be molded, greater draft angles will be required for release.

Mold Steels

Most steels used in plastic molds texture well. Many P-20, 0-1, A-6, S-7, H-13, and 420 stainless steel cavities have been textured successfully. It is imperative to use the same cavity material and have the texturing completed at the same time for textured parts that will be joined later in assembly. The texturing of cavities is useful in concealing flow lines, sink marks, and gate blushes on molded parts, in addition to being aesthetically pleasing.

Conditions Required for Polishing

Cleanliness is a crucial factor in improving the productivity of the polishing department, and standards should apply both to the workpiece and to the surrounding area. Between each polishing step, the workpiece must be cleaned thoroughly with soft tissue, soft rags, or a soft brush, and kerosene. An effective tool for keeping the workpiece free of metal particles and dust is a vacuum cleaner—as long as its brush is not used for general cleaning and is kept free of dust. Kerosene may also be used to thin the diamond compound when it dries.

In each step of polishing, the *lay* (direction of polishing) should be chosen. All traces of the previous lay must be completely removed before the next step is begun. Changing the lay makes it very easy to determine visually when the marks from the previous step have been completely removed.

The best polishing units are the flexible-shaft units. These incorporate $\frac{1}{10}$-hp universal motors with speeds up to 14,000 rpm. The speed control can be of the rheostat, carbon-pile, or electronic type. A standard shaft is 39 in. long, and a standard handpiece has a chuck that can accommodate shanks up to $\frac{5}{32}$ in. in diameter.

The first polishing step is a lapping operation, and a no. 15-5M blue diamond compound (12- to 22-μm range and a mesh of approximately 1,200) can be used. To hold the lap, which is round, $\frac{1}{2}$- and $\frac{3}{4}$-in. (1.27- and 1.91-cm) nylon bob holders are employed. Spring-loaded mandrels allow the handpiece to be slightly tilted and slight contours polished without fear of damage to the workpiece.

Short pieces of $\frac{1}{4}$- and $\frac{1}{2}$-in. (0.64- and 1.27-cm) brass nipples, with the outside cut down to fit the bobs, or round pieces of cast iron, with holes drilled through the centers, can be used as laps. The laps must fit snugly into the bob holders so that the handpiece can be lifted off the work without dropping the lap.

In areas inaccessible for lapping, a 600-grit silicon carbide stone may be used, but the work area has to be kept wet with kerosene. On areas that have been EDMed, a hard scale is left on the surface, resin-bonded stones do a better job than silicon carbide ones. If the EDMing was properly done, however, nothing coarser than a 600-grit stone should be used.

The next steps are all done with hard-bristle (not brass or steel-wire) brushes on $\frac{1}{8}$-in.-diameter mandrels. Successively finer-grit diamond, starting with no. 15-5M, then no. 6-48 M, 3-7M, and finally 1-8M, can be employed until the desired finish is achieved. A different brush should be used for each grade of compound.

The last step involves using a soft brush and the last grade of diamond compound used in the preceding step and covering the workpiece with a protective spray. Mold spray should also be used when the piece is left overnight, because it is not unusual for a piece of steel to start rusting within a matter of hours.

Sometimes, when a moldmaker has sent a mold in for polishing before it has been properly machine-finished, the polisher starts with

DNA Genotek Inc. Exhibit 2029 Page 159
Spectrum Solutions LLC v. DNA Genotek Inc. IPR2023-01424

stones as coarse as 100 grit in order to avoid sending the mold out for another setup. That is nearly always a serious mistake and will cost a lot of extra time.

When it is decided that the polisher must remove this excess roughness, he or she should use a no. 2 riffler or file instead of a stone. The only time a stone coarser than 600 grit should be used is when the mold has already been hardened. This itself is a very bad practice, since it is much easier to polish a soft mold.

Sometimes, an inexperienced polisher makes orange-peel and pit marks on the surface. These defects are caused by the metal flowing and tearing under heat and stresses beyond its elastic limit. The cure is to repeat the previous step.

The most reliable guide to efficient polishing is to use a grit half the size of that used in the previous step. For instance, if a piece has been polished with a no. 6 diamond compound, then the next one used should be a no. 3 diamond.

There is rarely any need for further polishing after the mold is properly heat-treated. The brown coating left by heat treating does not affect the finish, but it does give a lot of rust protection. Even though the shine may be gone, the finish is still the same, and no further work is needed.

As reviewed throughout this book, the complete operation has to be examined initially. Tight tolerances of all types can be met, including those for polishing, but at considerable cost. Unfortunately, some designers have a tendency to impose tight tolerances when they are not required. This is particularly true for polishing. Complex molds can be designed to permit a practical approach in polishing so that performance requirements of the molded part can be met at the lowest cost.

Platings, Coatings, and Heat Treatments

Different surface treatments are used to protect the processing tool (mold, etc.) against conditions such as abrasion and corrosion due to their contact with the melt. An example is physical vapor deposition (PVD) used to optimize the surface properties in a layer up to 10 mm deep that has little effect on the contour of part. PVD coatings lead to an insignificant though usually discernible roughening of the surface. Tools that have been machine-finished can therefore be improved. No expensive posttreatment is necessary. In this vacuum-chamber [10^{-2} to 10^{-4} mbar (1 to 0.01 Pa)] process, metals are converted to a gaseous state by the introduction of thermal energy (electron beam or arc) or kinetic energy (atomization). They condense on the surface being coated.

When molding corrosive reacting plastics such as CPVC parts of the mold cavity, cores, insets, etc. (also in dies), the metal surfaces can be subjected to corrosion and pitting. Certain steels such as stainless steel can provide a degree of protection. Hard coatings such as PVD TiN, TiCN, TiAiN, etc. are extremely corrosion-resistant and provide excellent abrasion resistance. They also enhance molded-part release. Different coating systems are used to protect the metal cavity's polished surface and/or extend its useful life, depending on the plastic being processed. They boost lubricity while avoiding melt adhesion problems.

More prevalent are chromium-based materials that can be applied at rather low temperatures to provide resistance to corrosion, abrasion, and/or erosion if needed. These include pure Cr, CrN (chromium nitride), and CrC (chromium carbide). Some coating methods such as PVD and chemical vapor deposition (CVD) subject the mold steels to excessively high temperatures that reduce steel hardness. Systems have been developed with PVD and CVD [plasma CVD (PCVD)] that operate at lower temperatures of 200 to 400°C (392 to 752°F). Popular is the so-called nitrided coating; it is actually a hardened nitride casing (nitrogen is absorbed into the surface of the steel).

Molders can be faced with sweating (moisture condensation) on their chilled mold surfaces, particularly during the summer months. This can lead to corrosion and rust, and in turn to poor finishes and inferior parts. In addition, rust on guide pins can cause damage

DNA Genotek Inc. Exhibit 2029 Page 160
Spectrum Solutions LLC v. DNA Genotek Inc. IPR2023-01424

Table 4-24 Mold surface plating and coating treatments

Process	Material	Applied to	Purpose
Coating by impingement, molecularly bound	Tungsten disulfide	Any metal	Reduce friction and metal-to-metal wear with dry film; nonmigrating
Coating by impingement, organically bound	Graphite	Any metal	Reduce sticking of plastics to mold surface; can migrate
Electrolyte plating	Hard chrome	Steel, nickel, copper alloys	Protect polish, reduce wear and corrosion (except for chlorine or fluorine plastics)
	Gold	Nickel, brass	Corrosion only
	Nickel	Steel and copper alloys	Resist corrosion except sulfur-bearing compounds, improve bond under chrome, build up and repair worn or undersized molds
Electroless plating	Nickel	Steel	Protect nonmolding surfaces from rusting
	Phosphor nickel	Steel and copper	Resist wear and corrosion
Nitriding	Nitrogen gas or ammonia	Certain steel alloys	Improve corrosion resistance, reduce wear and galling; alternative to chrome and nickel plating
Liquid nitriding	Patented bath	All ferrous alloys	Improve lubricity and minimize galling
Anodizing	Electrolytic oxidizing	Aluminum	Harden surface, improve wear and corrosion resistance

to the mold. By keeping the air in the plant, or around the mold, dry, you can not only improve part quality, but also increase your production rate.

Surface treatments used on molds are generally plated, coated, and/or heat-treated to resist wear, corrosion, and release problems. Treatments such as those reviewed in Table 4-24 that reduce wear are especially helpful in gates, runners, ejector pins, core pins, inserts, and cavity areas opposite the gate. Other treatments resist the corrosion damage inflicted by chemicals such as hydrochloric acid when processing PVC, ammonia with acetals, and oxidation caused by interaction between molds and moisture in the plant atmosphere. Release problems require treatments that decrease friction and increase lubricity in mold cavities.

No single mold treatment is ideal for solving all these problems. The molder must determine which mold problems are causing the greatest loss of productivity (or could cause loss) and then select the mold treatments that will be most effective in solving the problems.

Plating and coating affect only the surface of a mold or component, while heat-treating generally will affect the physical properties of the entire mold. Treatments such as carburizing and nitriding are considered to be surface treatments, and although heat is applied in these processes, they are not considered to be heat treatments. Heat-treating is more often the province of the steel manufacturer and moldmaker than of the molder. However, stress relieving is a heat treatment that the molder can perform.

Some mold wear cannot be prevented. This wear should be observed, acknowledged, and dealt with at intervals in the mold's useful life; otherwise, the mold might be allowed to wear past the point of economical repair. Periodic checks of how platings and coatings are holding up will allow the molder to have

DNA Genotek Inc. Exhibit 2029 Page 161
Spectrum Solutions LLC v. DNA Genotek Inc. IPR2023-01424

a mold resurfaced before damage is done to the substrate.

When a poorly finished mold is being used for the first time, its surface is actually being reworked by heat, pressure, and plastic. Fragmented metal is pulled out of the metal fissures, and plastic forced into them. While the fissures are plugged with plastic, the molder may actually be molding plastic against plastic. Breaking in a poorly finished mold can be haphazard without proper presurfacing. If the underlying mold surface is unsound (and no prior treatment was used although required), a thin layer of metal plating, particularly chrome plating, will not make it sound. A poorly prepared surface makes for poor adhesion between the treatment and the base metal.

Most molders or moldmakers will not find it practical to do very much surface treatment for themselves. Few of the plating, coating, and lubricating treatments lend themselves to being done in a molding shop. With few exceptions, treatments involve processes and chemicals that should not be used anywhere near an injection machine (because of corrosiveness), and they are best handled by custom plating and treating shops that specialize in their use.

The distinction between platings and coatings is not entirely clear. Generally, thin layers of metals applied to the surface of mold components are considered platings. The application of alloys, fluorocarbons, or fluoropolymers (such as TFE) or dry lubricants is considered a coating.

The effectiveness of a surface treatment depends on not only the material being applied, but also the process by which it is applied. For any plating or coating to stick to the surface of a mold component, it has to bond to the surface some way. The bonding may be relatively superficial, or it may be accomplished by a chemical (molecular) bond. The nature and strength of the bond directly affect the endurance and wear characteristics of the plating or coating. The experience of the plater is an important factor in applications where cut-and-dried or standard procedures have not been developed.

Nickel

Electroless nickel plating deposits up to 0.001 in. of nickel uniformly on all properly prepared surfaces. The term electroless is used to describe a chemical composition–deposition process that does not use electrodes to accomplish the plating. Plating by the use of electrodes is called electrodeposition. Electroless nickel provides a good protective treatment for mold components, including the holes for the heating media. It prevents corrosion; also, any steel surface that is exposed to water, PVC, or other corrosive materials or fumes benefits from this minimal plating protection. Electroless nickel has the characteristic of depositing to the same depth on all surfaces, which eliminates many of the problems associated with other metallic platings. Grooves, slots, and blind holes will receive the same thickness of plating as the rest of the part. This allows close tolerances to be maintained.

The surface hardness of electroless nickel is 48 Rockwell C, and the hardness can be increased by baking to 68 Rockwell C. Plated components will withstand temperatures of 700°F (371°C).

Chrome

Cromium is a hard, brittle, tensile-stressed metal that has good corrosion resistance on most materials. As it becomes thicker, it develops a pattern of tiny cracks because the stresses become greater than the strength of the coating. These cracks form a pattern that interlaces and sometimes extends to the base metal. A corrosive liquid or gas could penetrate to the base metal. This action can be prevented in three ways: A nickel undercoat can be applied to provide a corrosion-resistant barrier; the chrome plating can be applied only to a maximum thickness; or a thin dense chrome can be substituted.

Hard chrome can be deposited in a rather broad range of hardnesses, depending on plating-bath parameters. Average hardness is in the range of 66 to 70 Rockwell C. A deposit

DNA Genotek Inc. Exhibit 2029 Page 162
Spectrum Solutions LLC v. DNA Genotek Inc. IPR2023-01424

of over 0.001 in. (0.0025 cm) thickness is essential before chromium will assume its true hardness characteristics when used over unhardened base metals. Over a hardened base, however, this thickness is not necessary because of the substantial backing provided. Precise control of thickness tolerances can be achieved in a particular type of chrome plating generally referred to as thin, dense chrome. This kind of plating provides excellent resistance to abrasion, erosion, galling, cavitation, and corrosion wear.

Adhesion between a chrome layer and the base metal is achieved by a molecular bond. The bond strength is less on highly alloyed steels and on some nonferrous metals; however, bonds in excess of 35,000 psi tensile strength are common. Although it is often said that the chromium layer will reproduce faithfully every defect in the base material, actually, as the chrome deposit thickens, it will level imperfections. For smooth chrome plating, the base material should be at least as smooth as the expected chrome. Imperfections can be ground or polished after plating to erase them.

In some instances, cracks in the base material that are not visible through normal inspection techniques may become apparent only after plating. This phenomenon is attributed to the fact that grinding of steel often causes a surface flow of material, which spreads over cracks and flaws. However, this cover is dissolved during the preplating treatment, and the cracks become apparent as the coating thickness increases. The deposited layer of chromium, although extremely thick, will not bridge a large crack.

With electrodeposition, the current distribution over different areas of a component greatly varies, depending on its geometrical shape. Elevations and peaks, as well as areas directly facing the anodes, receive a higher current density than depressions, recesses, and areas away from the anodes that do not directly face the anodes. The variation in current density over different areas produces a corresponding variation in the thickness of the deposited metal.

Hard chrome plating is recommended to protect polished surfaces against scuffing and to provide a smooth release surface that will minimize sticking of the parts in the mold. Some precautions are necessary. Hydrochloric acid created in the injection molding of PVC will attack chrome. Chrome that is stressed and cracked under adverse conditions will permit erosion from water and gas penetration into the steel. To deal with hydrogen embrittlement created when hydrogen is absorbed by steel during the plating process, chrome-plated components should be stress-relieved within a half hour of completion of the plating. To protect against galling, chrome should not be permitted to rub against chrome or nickel.

Although it is not uncommon for a mold to be sampled in the machine under pressure before plating, this is an undesirable practice. The effects of moisture on the steel can cause chrome plating to strip later. Carelessness can also result in scratching an unplated mold. Dimensional checks can readily be made outside the press, using wax or other sampling materials. Chrome can be stripped from a mold after sampling in order to make essential changes. Periodic checks after a chrome-plated mold is in full production are desirable to find evidence of wear, which will show up first in the mold corners and high flow areas. A simple check can be made by swabbing a copper sulfate solution in the mold areas. If the copper starts to form a plating on the surface, the chrome is gone and must be replaced.

Nitriding and Carburizing

Steels also are nitrided and carburized to improve the surface hardness, thus making the surface more wear-resistant. Nitriding will penetrate the surface from 0.003 to 0.020 in. (0.008 to 0.051 cm), depending on the steel, and can result in hardnesses of 65 to 70 Rc. Another process for imparting a surface hardness is carburizing. In this process, carbon is introduced into the surface of the cavity or core steel, and the inserts are heated to above the steel's transformation temperature range while in contact with a carbonaceous material. This process frequently is

DNA Genotek Inc. Exhibit 2029 Page 163
Spectrum Solutions LLC v. DNA Genotek Inc. IPR2023-01424

followed by a quenching operation to impart the hardness case. Hardness as high as 64 Rc and a depth of 0.030 in. are possible with this process.

Other Plating Treatments

Numerous metals other than chromium and nickel have been used at one time or another to coat the components of plastics molds. Gold plating can be used to create a protective surface for PVC and some fluorocarbon materials. It prevents tarnishing, discoloration, and oxidation of the mold surface. Gold will protect the original finish and provide a hydrogen barrier. Fifty millionths of an inch of gold plating [0.00005 in. (0.0013 cm)] is adequate or the purpose. Gold can also be used as a primer under polished chrome. Platinum and silver have also been used to plate molds, and they share with gold the notable drawback of high cost.

Coating Treatments

Composites of ultrahard titanium carbides distributed throughout a steel or alloy matrix are used very effectively for coating. The carbides are very fine and smooth, and the coatings are applied by sintering, a process in which the component to be coated is preheated to sintering temperature, then immersed in the coating powder, withdrawn, and heated to a higher temperature to fuse the sintered coating to the component.

Flexibility in selecting and controlling the composition of the matrix alloy makes it possible to tailor the qualities of the alloy according to the requirements of the application. When heat-treatable-matrix alloys are used, the composite can be annealed and heat-treated, permitting conventional machining before hardening to 55 to 70 Rockwell C.

Standard alloy matrices have been developed with quench-hardenable tool steel, high-chromium stainless steels, high-nickel alloys, and age-hardenable alloys. Special alloys can be formulated for even the most corrosive conditions.

These types of coatings are effective in combating the severe wear that occurs with abrasive plastic compounds. Ceramic–metal composites provide the hardness and abrasion resistance to withstand wear by the most damaging glass-fiber-, mineral-, or ferrite-filled compounds. With the right metal-matrix selection, resistance to corrosion and heat is also obtained.

Treatments are available via a chemical process that utilizes thermal expansion and contraction to lock PTFE (fluoropolymer) particles into a hard electrodeposited surface such as chromium. Surfaces treated by the process are reported to have the sliding, low-friction, nonstick properties of PTFE, along with the hardness, thermal conductivity, and damage resistance of chrome. Core rods and other internal mold components benefit from this surface treatment. The process builds a thickness of 0.002 in. of electrodeposited chromium on component surfaces and is available for ferrous, copper, and aluminum-alloy parts (Table 4-25).

Impregnation processes are available that provide continuous lubrication to metal parts by impregnating fluoropolymers into the surface pores of the metal. Reduced friction, wear, and corrosion are reported as benefits, along with improved plastics flow and release characteristics.

Impregnation is applied to mold cavities, runners, core pins, and ejector pins, enabling easier ejection of parts and minimizing the need for release agents for most hard-setting thermoplastics. Pin galling and metal-to-metal sliding friction on core and ejector pins are also eliminated.

Titanium carbonitride is the material most commonly applied to provide wear resistance to tools and wear parts made from tool steels. The coating can often be applied in a layer thick enough to allow a stock for a surface-finishing operation after coating. The coating inhibits galling and results in a favorable coefficient of friction. The process protects against wear from abrasive fillers and corrosion from unreacted polymers that release acids. The coating is said to be 99% dense, that is, to have virtually no porosity, and is inert to acids.

DNA Genotek Inc. Exhibit 2029 Page 164
Spectrum Solutions LLC v. DNA Genotek Inc. IPR2023-01424

Table 4-25 Coatings

Material	Method of application
Chromium	Plating
Nickel	Plating
Electroless nickel	Solution treatment
Nedox electroless nickel	Solution treatment followed by TFE impregnation; used on copper and ferrous alloys
Tufram TFE aluminum	Deep anodizing process followed by TFE impregnation; used on aluminum alloys
TFE ceramic	Spray and bake application; used for all die materials that can withstand 250°C bake
Tungsten silicide	Solution treatment; used on steel and ferrous alloys
Tungsten carbide	Explosion impact or flame spray with plasma arc; used for all high-melting metals to improve abrasion resistance
Aluminum oxide	Plasma flame spray; used for extreme abrasion resistance; used on steel dies but usually limited to small dies because of expansion problems; works best on 18-8 stainless
PTFE	Spray and bake application; used for low-friction and low-adhesion application; poor abrasion resistance
Polyimide, aramid	Straight organic coatings with high softening points (450 to 500°C), which are applied by spray and baked; low-friction characteristics against some resins (e.g., PVC); moderate abrasion resistance
Filled polyimide, aramid	Aramid and polyimide systems containing TFE and other fluorocarbon resins to improve the friction properties

Note that the processes used for hardening screws and barrels are not necessarily the same as those applicable to mold components. A spin-casting process is used to coat the inside of barrels under heat, after which the barrels are rebored. The outsides of screws are hardened with gas or ion nitriding. Screws are also hardened by plasma-welding a bead of stellite on the screw surface, which is then ground to the desired shape and finish. The processes used on mold components are different in part because brittle tool steel cannot be twisted like a screw.

Heat Treatments

Most mold steels are subjected to heat treatment in some form to obtain the hardness necessary for their intended use. The routine heat-treating that is part of the mold-making process is not normally performed by molders, unless they also run their own mold-making shop.

Exceptional wear resistance of mold surfaces can be accomplished by carburizing or nitriding; both processes are also known as case hardening, which refers to the fact that the surface layer of the material being treated is made considerably harder than the interior. The depth of penetration of the treatments can vary from a few thousandths up to one-sixteenth of an inch.

Carburizing is accomplished by heating steel to between 1,600 and 1,850°F (871 and 1,010°C) in the presence of a solid carbonaceous material, a carbon-rich atmosphere, or liquid salts. Nitriding consists of subjecting parts to the action of ammonia gas at temperatures of 950 to 1,000°F (510 to 538°C) or to contact with nitrogenous materials in order to impregnate the surface with nitrogen.

These treatments can produce skin hardnesses considerably above the maximum hardness obtainable in heat-treated tool steels and provide excellent resistance to abrasion. General guidelines for heat-treating are readily available, but specific

DNA Genotek Inc. Exhibit 2029 Page 165
Spectrum Solutions LLC v. DNA Genotek Inc. IPR2023-01424

heat-treating data must always be obtained from the supplier of a particular grade of steel.

Anyone dealing with mold steels should be aware of potential sources of stress that can be detrimental to the life and usefulness of the finished tool. These sources of stresses may be a result of tool design, heat-treating, machining, grinding, EDMing, welding or brazing, or anything else that contributes to heating of the steel in a nonuniform manner. Heat-treating a mold always introduces risks of distortion and cracking; permanent linear movements in steel during heat treatment are to be expected. It is impossible to predict accurately the extent or direction of movement, since chemical composition, mass, geometry, design, and heat-treating techniques all affect the final dimensions of a mold.

Carburizing temperatures and hardening temperatures, which vary with the type of steel being used, are provided in technical data furnished by steel producers. Care must be taken during heat-treating to protect the mold surface against oxidation. This is done by packing the mold into spent cast-iron chips or pitch coke, heating in a controlled-atmosphere furnace, or heating in a vacuum furnace.

After the mold is heated to the hardening temperature, it is either quenched in a liquid such as oil or allowed to cool in air, depending on the analysis of the steel. High-alloy steels harden sufficiently when cooled in air from the hardening temperature.

A mold can benefit greatly from periodic stress relieving, a form of heat treatment highly recommended by metallurgists. This process can extend the life of questionable mold sections, even though they may not yet have exhibited cracks. The stress-relieving process consists of heating parts of the mold in question to the same temperature or just below the temperature at which the mold sections were tempered originally. The plating must be stripped before this annealing operation, since the stress-relief temperature is usually above the plating stability point.

Experience is the only available teacher to suggest the desired interval for this operation. Expensive mold sections clearly merit this form of preventive care, on the premise that an ounce of prevention is worth a pound of cure. Mold components with sharp or nearly sharp fillets and cores with a high ratio of length to cross section are vulnerable to cracking and therefore will require the most frequent stress relief.

There are practical limits to the use of mold treatments to solve molding problems. It makes more sense, for example, to replace $3 nitrided ejector pins than to spend great sums of money increasing their wear resistance further. Indeed, pins may break, no matter how much their surfaces are hardened, more often than they wear out. Furthermore, a pin nitrided to 70 Rockwell C will cause the hole around it to wear before the pin does. The solution to that problem is to ream the hole out and install an oversize pin. Judicious use of the various forms of plating, coating, and heat-treating processes available to today's molder will go a long way toward maximizing mold life and productivity. Well-treated molds, closely observed for wear and diligently maintained, are likely to provide the molder with fast, long, and profitable operation.

Cleaning Molds and Machine Parts

Overview

Removal of residual plastics from molds and machine parts by pulling off or mechanical separation with a knife is the easiest method of cleaning a mold. In most cases, mechanical cleaning is confined to uncomplicated molds (or screws) and the use of no-problem plastics. As soon as surface conditioning by chemical or thermal posttreatment is necessary, the relevant regulations against air and water pollution and for accident prevention, as well as the applicable threshold limit values, have to be observed. All plastics, both thermoplastics and thermosets, can be thermally moved.

Thermogravimetry provides graphs plotting the decomposition of plastics. TPs

DNA Genotek Inc. Exhibit 2029 Page 166
Spectrum Solutions LLC v. DNA Genotek Inc. IPR2023-01424

Table 4-26 Cleaning-method guide

Cleaning Process	Manual	Oven	Solvent	Triethylene Glycol	Post	Salt Bath	Ultrasonic	Fluidized Bed	Vacuum Pyrolysis
Required auxiliaries	Acetylene and oxygen		Cooling water, ethylene glycol	Cooling water, triethylene glycol, or polymer-specific solvent	HNO_3, water	$NaNO_2$ or 10% NaOH + 45% $NaNO_2$ + 45% KNO_3 or similar	Water, wetting agent	Al_2O_3 sand	Water
Cleaning temperatures, °C	Locally to 600	400–500	280–285	280–285	40–90	320–300	20–70	400–550	360–525
Total cleaning time, dwell time in cleaning oven, h		8–16; 6–12	8–10; 4–6	8–10; 4–6	2–4; 1–3	4–6; 4–6	1–2; 1–2	3–4; 2–3	5–7; 4–5
Recommended number of cycles		1	2	2	1	1	1	1	1
Postcleanings		Labor-intensive	1 to 2 baths, ultrasonic	1 to 2 baths, ultrasonic	—	3 baths, ultrasonic	—	Ultrasonic	Ultrasonic
Waste products	Burnt plastics	Burnt plastics ash	Contaminated ethylene glycol	Contaminated triethylene glycol	Acids and salts	Contaminated salt	—	Contaminated Al_2O_3 ash	Contaminated plastic

DNA Genotek Inc. Exhibit 2029 Page 167
Spectrum Solutions LLC v. DNA Genotek Inc. IPR2023-01424

Environmental pollution	Heavy	Very heavy due to soot and combustion gases	Very low: contaminated triethylene glycol, reprocessed through distillation	Very low: contaminated triethylene glycol, reprocessed through distillation	Heavy	Very heavy due to soot and combustion gases	—	Very heavy due to soot and combustion gases, moderate if postburning is used	Hardly any	
Cleaning effect	Dependent on personnel	Brushoff, postclean, carbonized residues	Very clean, including interior parts without dismounting, very gentle, carbonized polymers are not dissolved		Very good as postcleaning	Very good; wash off salt crusts and neutralize	Very good as postcleaning	Very good externally	Very good with following final cleaning stage	
Suitable for	Only as auxiliary	All steel and stainless steel parts contaminated with plastics	Multilayer and sinter metal filters, vanadium sand, mold interior, also aluminum parts, only for removal of PET and PA		Stainless steel parts	All steel and stainless steel parts contaminated with plastics	Final cleaning after all processes	All steel and stainless steel parts contaminated with plastics	Steel and vanadium parts, filters resistant to 400°C gaps of 0.2-mm diameter	
Not suitable for		Multilayer filters	Other polymers requiring specific solvents		Steel, bronze, aluminum	Sintered metal, sintered web, aluminum		Sintered metal, sintered glass, aluminum filter	Open aluminum parts, soldered parts	
Cleaning cost and labor index (10 highest)	10	6	7	$6\frac{1}{2}$	1	6	1	$6\frac{1}{2}$	4	

DNA Genotek Inc. Exhibit 2029 Page 168
Spectrum Solutions LLC v. DNA Genotek Inc. IPR2023-01424

disintegrate at temperatures between 300 and 400°C, a 100% loss of weight occurring within a very short period of time. With TSs, weight loss, characteristic of the decomposition process, can start between 300 and 400°C (98).

A variety of methods are used to carry out thermal decomposition in mold cleaning (Table 4-26). The conventional methods of flaming or treatment of the molds in oven chambers have to take into account pollutants that occur. Substantial overheating can occur when flammable components of plastics burn up in an oxygen-containing atmosphere upon reaching their flash point. Mold damage is frequently attributed to these rather uncontrollable reactions of exothermic combustion. Even authentic chrome–nickel steel grades may be at risk under such conditions due to carbonization.

Open solvent baths are used for mold cleaning. Solvents used in closed systems are only appropriate under economic conditions where the required cleaning results are not obtainable with any other method. Some molds may justify the use of solvents; however, cleaning in an oil bath yields equally good results in most cases. Solvents should be selected individually for the various plastics. Losses due to evaporation and reprocessing of at least 10% must be expected even with closed systems.

Cleaning in an oil bath involves the immersing and heating of the molds within a closed container. The oil used can consist of a synthetic mixture of isomeric dibenzyl toluenes. The equipment used consists of electrically heatable stainless steel tanks adapted to the mold geometries. To reduce the oil oxidation and improve aging behavior, the cleaning process is carried out under a nitrogen blanket.

Manual Cleaning

Cleaning of machine parts by hand, either by mechanical removal of the hardened plastic layers or melting off with the aid of an acetylene torch, is the most elaborate method, but is still practiced in many small firms. Mechanical damage and/or local heat deformation of the mold are the risks of this method. In addition, the quality of cleaning is very dependent on the skill and reliability of the workers carrying it out. If we consider the high wage costs and secondary costs, this is the most expensive method, but it requires almost no investment.

Oven Cleaning

Both vacuum and conventional ovens are still in general use for pyrolysis-type cleaning. Generally, the advantages of ovens lie in their relatively low capital equipment cost and convenience in handling large loads. Ovens, however, operate on a very long cleaning cycle, and significant postcleaning operations are required to remove residual carbon. This is normally a manual operation on critical hardware. Glass bead cleaning is used on hardware in which precise maintenance or dimensional tolerance is nonessential.

Vacuum ovens tend to require high maintenance, especially on the pumps and seals, and in some cases, special provisions are made when handling certain plastics that require collecting vapors and/or residue released from the molds or dies (different types of filters or scrubbers are used).

In the case of both types of ovens, complex assembled hardware is normally broken down prior to cleaning. When the disassembly of parts is not possible, dual-cycle cleaning with disassembly between cycles is frequently required. This practice obviously increases both operating costs and turnaround time.

Finally, nonuniform heating creates stress and can be preferentially destructive to metallurgical properties induced by prior heat treatment. This consideration becomes more important with the increased use of precipitation-hardened stainless steels where dramatic changes in hardness occur over a relatively small temperature differential.

Solvent Cleaning

Solvent cleaning of hardware can be accomplished with acid or alkaline chemicals,

DNA Genotek Inc. Exhibit 2029 Page 169
Spectrum Solutions LLC v. DNA Genotek Inc. IPR2023-01424

such as ethylene glycol and organic agents, and by organic or inorganic ultrasonic cleaning. Conventional solvent baths require the complete breakdown of complex hardware prior to cleaning, often resulting in mechanical damage to the hardware. Acid or alkaline cleaning is slow at best and frequently corrosive to hardware. Although equipment costs are low, chemical costs are high. Sludge removal and disposal are both labor-sensitive and an increasing environmental problem. Organic solvents can be effective; however, environmental considerations restrict their use, and recycling equipment is necessary due to environmental pollution controls.

Triethylene Glycol Cleaning

This TEG method is very suitable for parts contaminated with polyester and nylon, since the boiling point of TEG is about 285°C at normal pressure, for example, in a water-cooled, but open, reflux condenser. The intensely bubbly bath removes the plastic even inside hollow cavities if they have inlet openings.

Postcleaning

A nitric acid postcleaning is necessary as a supplement to solvent cleaning processes if the carbonized plastics residues, additives, and pigments are not sufficiently removed from the workpieces. Depending on the parts to be cleaned and the adhering plastics, this may also be done by alkaline postcleaning in the same, or other similar, apparatus. Twenty-five to 30% strongly alkaline stripper is dissolved in water, and the parts to be cleaned are treated in this for approximately 2 h at up to 80°C. This type of acid or alkaline post-treatment must always be succeeded by neutralization in one or two stages, the last stage of which must be boiling out in pure water.

Salt Bath Cleaning

For a long time, molds to be cleaned have been immersed in baths of molten salt in which the plastic melts and burns. The simplest salt for this is molten sodium nitrite or mixtures of this with smaller quantities of other salts. Salt baths operate primarily on the basis of thermal oxidative decomposition of the polymer. There can be chemical corrosion associated with this cleaning method, and surface defects created by thermal shock during the water rinsing of salt from hardware at an elevated temperature. It is not normally considered for new or replacement installations, because of environmental and safety considerations. Other disadvantages include high operating costs (since the system must be left on even when not in use), high replacement costs of salt, and sludging and disposal problems associated with spent salt.

Ultrasonic Solvent Cleaning

Ultrasonic solvent cleaning will normally improve solvent bath cleaning dramatically. However, corrosion, disposal problems, and expensive chemical costs still remain. Also, equipment costs rise significantly when ultrasonic cleaning is required. It is used most successfully as a postcleaning process for the removal of inorganic residues from a thermal cleaning system.

Fluidized-Bed Cleaning

Fluidized-bed cleaning, introduced in the late 1960s, has for most applications become the option of choice. Absolute temperature control and thermal uniformity permit cleaning of complex critical hardware with minimal risk of distortion or metallurgical damage. Also, the average cleaning cycle is an order of magnitude shorter than that for ovens, due to the superior heat-transfer characteristics of the fluid bed.

Vacuum Pyrolysis Cleaning

Only vacuum pyrolysis provides good-quality, pollution-free cleaning of molds for almost all plastics, without additional chemicals. Apart from the vacuum pyrolysis oven,

DNA Genotek Inc. Exhibit 2029 Page 170
Spectrum Solutions LLC v. DNA Genotek Inc. IPR2023-01424

only electricity and water are used. The parts to be cleaned are laid in a basket or hung in a frame, with the largest opening directed downward. The loaded vacuum autoclave is evacuated to 50 to 10 mbar and heated initially to 50 to 100°C above the melting temperature of the plastic. More than 90% of the plastic runs from the molds into a vessel connected below, where the polymer hardens.

The actual cleaning process is for less than 10% of the originally adhering plastic. After removal of most of the plastic, the heater is adjusted to vacuum pyrolysis temperature: between 370 and 520°C depending on the plastic. When this temperature has been reached, the residues remaining on the mold surfaces are pyrolyzed under vacuum. The same occurs with the plastics residues inside the mold.

Strength Requirements for Molds

The forces involved with the molding operation are compressive; they are exerted by the clamping ram and internal melt pressure. Forces exist inside a cavity as a result of injecting the plastic material under pressure. If we inspect the stationary platens of injection machines in an operating plant, we will find that a number of them have indentations and impressions. These are a result of projections from the mold base and, in some cases, of the mold base being too small for the clamping force, thus causing a concentrated stress in the platen that brings about the flow of the platen metal. The platen impressions are dangerous because they reduce the contact area for the mold, thereby increasing the potential for further indentation. These indentations, if permitted to increase in number, may ultimately cause cracking of a platen, which would not only take the press out of operation, but also require a large and expensive replacement.

Practically all presses have provisions for reducing the clamp tonnage, but the problem is to recognize the danger and limits within which it is safe to concentrate a load on the platen. Most platens are made of cast steel with a yield strength of about 25 tons/sq in. Allowing a safety factor of 7, we have a permissible load of 3.5 tons/sq in. (0.54 tons/sq cm). With this information, we are able to calculate the minimum number of square inches a certain press size will safely accommodate or to determine to what tonnage to reduce the clamp in order to protect the platen against damage.

A mold 10 in. × 12 in. (25.4 × 30.5 cm) is to be placed in a 500-ton press. First, we will establish the minimum area of mold base needed to safely absorb the clamp force

$$500 \text{ tons} = \text{area} \times 3.5 \text{ (permissible stress)}$$

$$\text{area} = \frac{500}{3.5} = 143 \text{ sq in. } (923 \text{ sq cm})$$

The mold in question is 10 in. × 12 in. = 120 sq in. (774 sq cm). This shows that the mold should be operated with a reduced tonnage, calculated as follows:

$$\begin{aligned}\text{Tonnage} &= (\text{area} - \text{locating-hole area}) \\ &\quad \times 3.5 \text{ (permissible stress)} \\ &= (120 - 12.5) \times 3.5 \\ &= 107.5 \times 3.5 = 376 \text{ tons}\end{aligned}$$

To protect press platens against damage, it is advisable to check the contact area of the mold and platen to see that the safe permissible load is not exceeded.

Stress Level in Steel

When we examine the great variety and complexity of plastic parts, we realize that the molds in which they are produced are even more complex. A great many factors should be remembered when a design layout for a mold is made, but none is more important than maintaining a low stress level in the steel of all the components of the cavity and core. Highly stressed parts mean short tool life.

Heavy and high-speed cuts during metal removal, severe grinding action, and (to a much lesser degree) electric discharge machining all produce some amount of stress in various tool steels. Stress relieving will minimize the danger of failure.

Molds that are built for long life and high activity are heat-treated either initially or

DNA Genotek Inc. Exhibit 2029 Page 171
Spectrum Solutions LLC v. DNA Genotek Inc. IPR2023-01424

whenever the intermediate hardness of the cavities begins to affect the quality of the product unfavorably. From a heat-treatment standpoint, the tool designer should be on the lookout for the following:

1. The parts should be so shaped that they will heat and cool as uniformly as possible. A part that may heat so that a temperature difference exists between two points will produce a harmful strain when quenched.

2. A balanced section will heat and cool more uniformly than an unbalanced one, thus guaranteeing a much lower stress level.

3. Holes may be used to reduce the mass of metal in one area to offset the lower mass in an adjacent point.

4. Sharp angles and corners are a most common error that, with a little effort, could be minimized. Sharp corners and angles are points of high stress concentration. When a rectangular insert is being made for the cavity, the sharp corner in the plastic will most likely tolerate a radius of 0.020 in. (0.051 cm). If this is not permissible, the insert portion that is even with the cavity can be made larger and have a generous radius; the portion that is molding can be of whatever shape is required.

5. A thin section will cool faster than a thick one during quenching and will set up stresses. A larger radius or an even taper in the transition area will minimize stresses.

6. For whatever purpose they are intended, blind holes should be eliminated. A through hole makes for greater uniformity in cooling and eliminates the stress concentration from the sharp corner at the bottom of the hole. Junctions of holes, such as might be planned for fluid circulation, should be avoided in favor of drilled-through holes, since the intersection of holes will act to raise stress.

The designer should be aware that the best choice in material coupled with the best effort of the heat treater cannot overcome faulty design. When layouts for the cavity and core are made, an outline of the components should be presented to a heat treater for a recommendation of design modification that will lead to parts with low-level stresses. This matter deserves serious consideration. An order to the heat treater should specify: "To be stress-relieved if heat-treating steps will not reduce stress."

Pillar Supports

The construction of a mold base usually incorporates the U-shaped ejection housing. If the span between the arms of the U is long enough, the forces of injection can cause a sizable deflection in the plates that are supported by the ejector housing. Such a deflection will cause flashing of parts. To overcome this problem, the span between supports is reduced by placing pillar supports at certain spacings so that the deflection is negligible.

For the determination of pillars and their spacing, the beam formula can be applied. For this purpose, we consider a $1\frac{7}{8}$-in.-thick plate (Fig. 4-133) as beam-supported at a 8.5-in. centers with a uniform load. For this loading system, we consult a handbook (under "Stresses of Beams at the Center") to find the stress (7):

$$\text{Stress at center} = \frac{WL}{8Z} = S$$

where W = the load that the plate can support
L = the length between supports = 8.5 in. (Fig. 4-133)
Z = the section modulus (a property of the cross section that represents resistance to flexure)

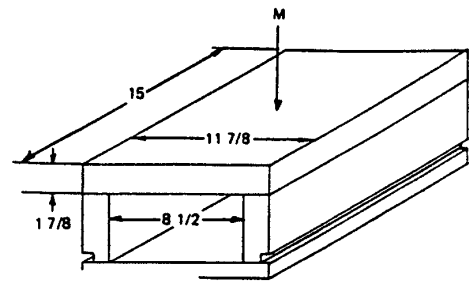


Fig. 4-133 Schematic for pillar requirements.

In the handbook (under "Section Modulus"), we find the following formula:

$$Z = \frac{bd^2}{6} = \frac{bB^2}{6} = \frac{15 \times 1.875^2}{6}$$

in which

$$d = B = 1.875$$
$$b = 15$$
$$Z = \frac{15 \times 3.52}{6} = 8.80$$

S is the allowable safe stress, and the value suggested by the mold base manufacturer is 12,000 psi (82.7 MPa). Referring to Fig. 4-133 and using the formula for S, we obtain

$$12{,}000 = \frac{W = 8.5}{8 \times 8.8}$$
$$W = \frac{12{,}000 \times 8 \times 8.8}{8.5}$$
$$= 99{,}275 \text{ lb} \quad (45{,}071 \text{ kg})$$

where W = the permissible load on the support plate

When the mold is closed, the cavities will exert a concentrated pressure on the support plate. For this condition, a safe concentrated stress in compression of 7,000 psi (48.3 MPa) is allowed. The compression formula from the handbook is

$$S = \frac{P}{A}$$

where S = 7,000 psi allowable stress
$P = W = 99{,}275$

Thus,

$$A = \frac{P}{S} = \frac{99{,}275}{7{,}000}$$
$$= 14.13 \text{ sq in.} \quad (91.16 \text{ sq cm})$$

Therefore, the total area of the back of the cavities can be only 14.13 sq in. If one row of support pillars is added, the dimension L is 4.25 in. (10.80 cm) in the load formula, thus doubling the load capacity of the plate and also the permissible cavity area to 28.26 sq in. (182.32 sq cm), and at the same time maintaining the allowable stress of 7,000 psi (48.2 MPa). Increasing the number of rows of pillar supports decreases the distance between the resting points of the beam, thereby increasing the area for the concentrated pressure of the cavities.

Steel and Size of Mold Base

The size and type of mold base are determined by placement of the cavities, the method of feeding the cavities, the ejection employed, the type of pockets desired, temperature control, type of cam action, or any unusual factor that becomes necessary for a specific part. A layout of these and the other elements so far established will indicate the type and overall size of the mold base.

Taking a four-cavity mold as an example, we obtain the outside width and length of the cavities. To the width of the cavities, we add 1.75 in. (4.45 cm) per side, so that they are placed close to support blocks of the ejector housing. On the end, we add an additional inch for the return pins, giving a total of 2.75 in. (6.99 cm) per end. These overall dimensions are checked against standard available mold bases, and the selection is made to satisfy the outline of the layout.

There are normally three grades of steel employed in mold bases, as follows:

1. The lowest-priced steel grade in a mold base is a medium carbon type with tensile strength of 55 to 75 $\times$ 10^3 psi (0.38 to 0.52$\times$ 10^3 MPa). This grade is suitable for application when the cavities are in themselves strong enough to withstand the conditions of application. The main function of a mold base in the preceding case is to keep the two halves aligned and the ejection side rigid enough to permit ease of ejection on a cycle of two or more times a minute. When the cavities are mounted in a cut-through plate, care must be exerted that the surrounding frame is thick and wide enough to safeguard the guiding features of the halves. When blind pockets are employed, this steel is suitable for a majority of applications.

2. The next higher grade of steel employed for bases is an AISI-type 4130, heat-treated to a hardness of 300 Bhn with a tensile strength of 126 to 155 $\times$ 10^3 psi (0.868 to 1.070 MPa).

DNA Genotek Inc. Exhibit 2029 Page 173
Spectrum Solutions LLC v. DNA Genotek Inc. IPR2023-01424

This grade is usually considered for cases where the cavities are constructed in sections, and it is the function of the plates to retain these sections without allowing them to separate under the forces of injection pressures. It is also applied in cases where cooling lines and other machining requirements weaken the cavity plate to a point whereby a material with higher physical properties is prescribed.

3. There are occasions when it is desirable to machine cavities into the cavity plates instead of fabricating cavities and inserting them into mold-base plates. This may be the case for a product with a yearly activity of less than 10,000 pieces and configuration that is relatively easy to machine from a mold-base plate. For such application, the mold-base cavity plates may be specified to be an AISI 4135 steel heat-treated to 300 Bhn with a physical strength of 129 to 155×10^3 psi (0.889 to 1.070 MPa). It is a suitable steel for polishing and higher-hardness heat-treating if necessary.

Deformation of Mold

The function of a mold is to receive molten plastic material from the plasticator (injection unit) ranging in temperature from 350 to 900°F (177 to 482°C) at pressures between 4,000 and 20,000 psi (27.6 to 137.8 MPa). In the injection process, the plastic comes from a heated nozzle and passes through a sprue bushing into feed lines (runners) to a gate into a cavity. The cavities are maintained at temperatures generally ranging from 30 to 350°F (−1 to 177°C) for thermoplastics and 250 to 600°F (121 to 316°C) for thermosets, at which solidification takes place. They are provided with a means for controlling the temperature.

At the end of the injection stroke, and during after-filling, pressure is built up in the mold cavity. This pressure, which depends on the type of molding and plastic, is generally one-third to one-half of the injection pressure set on the machine. In normal cases, the pressure in the mold cavity will be up to 4,000 to 8,000 psi (27.6 to 55.1 MPa). However, in exceptional instances it may rise to 15,000 psi (103.4 MPa) in certain mold components, usually when close dimensional tolerances need to be held. The consequences of such pressures must be appreciated. They cause elastic deformations, such as bending of cavity retainer plates and cores, that are virtually unavoidable.

The use of a sturdy construction (sufficiently thick cavity retainer plates, and support pillars in the open gap for the ejector system) may reduce elastic deformation to an acceptable level. Such possibilities are, however, often restricted, since light construction is required for efficient cooling, necessary spaces for guide pins and ejector system, etc. Elastic deformation of weak or insufficiently solid mold components may result in:

1. Differences in wall thickness with consequent excessive dimensional variations, as well as insufficient dimensional stability and rigidity of the molding.

2. Nonuniform melt flow in the mold. In the case of thin-walled moldings, this may give rise to flow lines, weld lines, internal stresses, or even trapped air.

3. In weak molds, the bearing surfaces or other components being forced apart by the plastic pressure, causing flash formation that may interfere with proper mold release. Moreover, the subsequent deflashing operation is a considerable cost-raising factor.

4. Faulty operation of the ejector system and guide pins. It is even possible that the mold will jam.

The general principles of molding are similar regardless of the type of press employed. All presses must meet the basic elements of molding: time, temperature, and pressure, the range of temperatures and pressures depending on the type of plastic material. The plastic is held in the cavity for a prescribed time until full solidification takes place; at this point, the mold opens, exposing the part to the ejection or removal action.

Mold Filling

The effect of mold dimensions and resin viscosity on pressure requirements is

DNA Genotek Inc. Exhibit 2029 Page 174
Spectrum Solutions LLC v. DNA Genotek Inc. IPR2023-01424

expressed as follows:

$$Q = \frac{p}{K\eta}$$

or

$$p = K\eta Q$$

where Q = volumetric rate of mold fill (cu in./sec or cu cm/sec)
p = pressure at mold entrance (lbf/sq in. or kgf/sq cm)
η = resin viscosity (lbf sec/sq in. or kgf sec/sq cm)
K = mold flow resistance factor

For the end-gated rectangular cavity section fillout,

$$K = \frac{12L}{W_t^3}$$

For the end-gated annular cavity section fillout,

$$K = \frac{12L}{\pi D_m t^3}$$

For the end-gated cylindrical cavity section fillout,

$$K = \frac{128(L + 4D_c)}{\pi D_c^4}$$

In these equations,

L = mold cavity length (in. or cm)
W = mold cavity width (in. or cm)
t = mold cavity thickness (in. or cm)
D_m = mean diameter of annulus mold cavity (in. or cm)
D_c = mean diameter of cylindrical mold cavity (in. or cm)

From the above, it may be seen that a constant flow rate and resin viscosity:

1. The pressure required to fill is directly proportional to the mold length.
2. The pressure required to fill is inversely proportional to cavity width or diameter.
3. The pressure required to fill is inversely proportional to the cube of the mold thickness.

The pressure required to fill radial fill patterns (i.e., center-gated) is exponential in the radius.

The pressure required is proportional to the resin viscosity and is reduced by an increase in temperature and/or shear rate as the following equations indicate:

Shear rate for rectangular section

$$= \frac{6Q}{t^3(W + t)} \text{ sec}^{-1}$$

Shear rate for annular section

$$= \frac{6Q}{t^2(\pi D_m + t)} \text{ sec}^{-1}$$

Shear rate for cylindrical section

$$= \frac{32Q}{\pi D_c^3} \text{ sec}^{-1}$$

Thus, shear rate is increased and resin viscosity decreased by a decrease in mold cavity dimensions (i.e., an effect opposite to that which such cavity dimensions have on the mold flow resistance factor K)

Deflection of Mold Side Walls

Rectangular cavities The maximum deflection commonly allowed in such molds is 0.005 to 0.01 in. (0.13 to 0.25 mm), depending on the size of the tool. Of this, 0.004 to 0.008 in. (0.1 to 0.2 mm) may be due to clearances between the blocks of the buildup mold and elongation of the bolster or register faces. For stress design purposes, therefore, a maximum deflection of 0.001 to 0.002 in. (0.025 to 0.05 mm) is usually taken. The approximate thickness of the side wall required may be calculated from the following formulas:

$$y = \frac{Cpd^4}{Et^3}$$

or

$$t = \sqrt[3]{\frac{Cpd^4}{Ey}}$$

where y = deflection of side walls (in. or cm)
C = constant (see Table 4-27)
p = maximum cavity pressure (lbf/sq in. or kgf/sq cm)
d = total depth of cavity wall (in. or cm)

DNA Genotek Inc. Exhibit 2029 Page 175
Spectrum Solutions LLC v. DNA Genotek Inc. IPR2023-01424

Table 4-27 Constant C used in calculating deflection of mold side walls

Ratio of Length to Depth of Cavity Wall	C
1 : 1	0.044
2 : 1	0.111
3 : 1	0.134
4 : 1	0.140
5 : 1	0.142

E = modulus of elasticity for steel (30×10^6 lbf/sq in. or 2.1×10^6 kgf/sq cm)
t = thickness of cavity wall (in. or cm)

Cylindrical cavities The increase in radius due to the internal pressure of the injected material can be determined approximately as follows:

$$r_1 = \frac{rp}{E}\{[(R^2 + r^2)(R^2 - r^2)] + m\}$$

where r_1 = increase of radius (in. or cm)
r = original inside radius (in. or cm)
R = original outside radius (in. or cm)
m = Poisson's ratio (0.25 for steel)

The strength requirements for the two configurations are satisfactorily met. In all the calculations, it was taken for granted that the ram pressure was applied to the cavities only. This was accomplished by having the cavity insert protrude above the A or B plate about 0.005 in. (0.013 cm).

Let us now assume that for some valid reason, a two-cavity mold is ordered, and the press in which it is to be run is still the 200-ton size. In this case, the width of the cavity face would be unchanged except that cavity inserts would be mounted flush with A and B plates so that the plates would absorb part of the force.

The problem of mounting a cavity will be favorably met in either a machined-through picture-frame pocket or blind pocket, whichever is more suitable from the standpoint of mold temperature control, as well as other considerations. Based purely on strength considerations, the calculated dimensions will incorporate in the cavity itself the ability to safely absorb all the forces to which it may be subjected during molding.

During injection of the fluid plastic into the cavity, we find pressures existing there between 4,000 and 10,000 psi (27.6 to 69 MPa) close to the point of exit from the gate. As the flow approaches the outside extreme point, these pressures may be 2,000 psi (13.8 MPa). The difference in readings between those of the pressure gauge and 2,000 psi at the end of flow is found in the pressure drops coming from the screw acting as a plunger, the nozzle, the sprue bushing, the runners, the gate, and resistance to flow within the cavity. The average pressure in the cavity may be 4,000 to 10,000 psi. Even the lower pressures in a cavity will cause a sizable deflection in a cavity wall unless it is made heavy enough to keep such deflection within acceptable values. The following takes place in the cavity: The projected area of the side wall times the pressure in the cavity creates a force that will bring about a movement of the side wall of, say, 0.003 in. (0.0076 cm). After the material is cooled and the inside pressure drops to a negligible value, we have a force from the deflected steel tending to return to zero deflection. This force is comparable to that which caused the original deflection.

If the part is made of a thickness that will shrink 0.003 in. in the cavity, then the steel will merely go back to its original position without any ill effects on the operation. If, however, the plastic will only shrink 0.001 in. (0.0025 cm), the steel pushes into the plastic by 0.002 in. (0.0051 cm), causing difficulty in mold opening, possibly marring the surface, and adversely influencing the dimensions and properties of the plastic. Last but not least, the large forces involved will gradually cause the movement of cores with respect to cavities, with additional complications and ultimately mold damage.

The problem of mold deflection must be solved in a way that will eliminate these difficulties. A cavity must be looked on as a very high-pressure vessel in these considerations. Some of the formulas related to the subject are found in Raymond J. Rark and Warren G.

DNA Genotek Inc. Exhibit 2029 Page 176
Spectrum Solutions LLC v. DNA Genotek Inc. IPR2023-01424

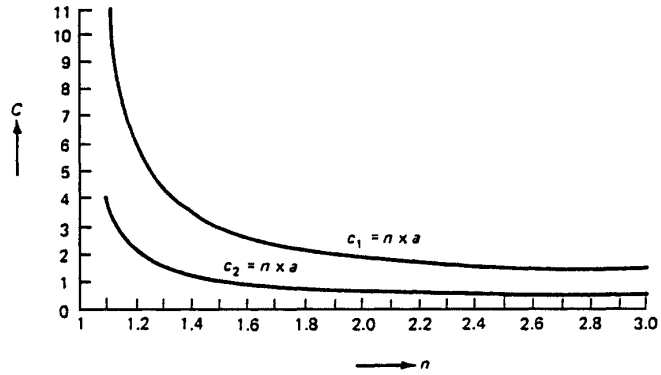


Fig. 4-134 Factors to use in determining deflection in a mold cavity.

Young, *Formulas for Stress and Strain* (New York: McGraw Hill, 1975).

These formulas have been modified and rearranged here to suit conditions that exist in molds. The formula for deflection in a cavity *without* the restraining effect of top and/or bottom is

$$d_1 = \frac{pac_1}{E} = \text{deflection}$$

A cavity *with* restraining effect in which the bottom is either an integral part of the cavity or is so interlocked that it will act as an integral part of the unit shows this deflection:

$$d_2 = \frac{pac_2}{E} = \text{deflection}$$

where p = average pressure in the cavity (psi)
a = half the width of a part as viewed from the top
c_1 and c_2 = factors from Fig. 4-134 (equal to $n \times a$)
E = modulus of elasticity for steel [30×10^6 psi (0.21×10^6 MPa)]

The factor c_2 is applicable only in cases when the depth of the cavity is equal to a. For greater depths, there is a gradual transition to the deflection that exists in the case of an unrestrained bottom or top. The distance at which the c condition will be reached is

$$L = a\sqrt{\frac{\text{unstricted deflection}}{\text{restrained deflection}}}$$

In practice, the requirements for creating equivalent conditions that will correspond to an integral solid bottom or top are:

1. The cavity insert must have metal-to-metal contact with its retainer; that is, the outer diameter of the insert and inner diameter of the retainer must be exactly the same and, when assembled, result in a light press fit.

2. The body of the cavity and bottom or top must be so interlocked that there is no chance that the insert will move with respect to its retainer.

3. The clamping pressure should be 25% higher than would be the case if there were no deflection problem.

Let us take an example: The average cavity pressure is 6,000 psi (41.4 MPa) and the construction will be of the restraining type, and we suppose

Part depth = 8.5 in. (21.6 cm)
Average diameter = 8.0 in. (20.3 cm)
Wall thickness = 0.070 in. (0.178 cm)
Shrinkage = 0.020 in./in.

The total shrinkage on the part will be about 0.0014 in. (0.0036 cm), but when the pressure in the cavity is decaying, it may be only half that amount, or 0.0007 in. (0.0018 cm). If we allow for a maximum deflection in the center of the cavity of 0.0006 in. (0.0015 cm), there should be no ill effects on part performance. If the cavity is constructed of sections, no plastic material will flow between sections at selected deflection. Thus, the mold should work properly.

DNA Genotek Inc. Exhibit 2029 Page 177
Spectrum Solutions LLC v. DNA Genotek Inc. IPR2023-01424

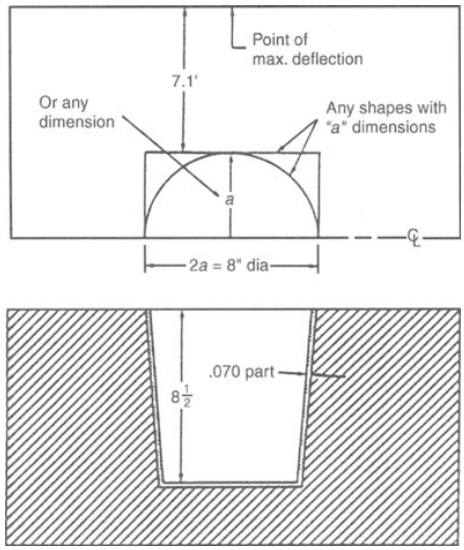


Fig. 4-135 Cavity for a product when average pressure is 6,000 psi (41 MPa).

Therefore,

$$d_2 = \frac{pac_2}{E} = \frac{6{,}000 \times 4 \times c_2}{30 \times 10^6} = 0.0006$$

$$c_2 = \frac{30 \times 10^6 \times 0.0006}{6{,}000 \times 4} = \frac{18}{24} = 0.75$$

From the curve in Fig. 4-134 we find $n = 2.4$. The outside cavity dimension will be $n \times a = 2.4 \times 4 = 9.6$, giving a wall of 5.6 in. (14.2 cm). Since water lines will be incorporated and cause a decrease in strength, we will add $1\frac{1}{2}$ in. (3.81 cm) for 1-in. baffled lines and $\frac{3}{4}$ in. for $\frac{7}{16}$-in. lines. In this case, the addition will be $1\frac{1}{2}$ in., and new thickness 7.1 in. This will be satisfactory for the 4-in. depth of cavity, and since the cavity is 8 in. in diameter, we will arrange the core end of the mold for interlocking and restricting action. Thus, the 4-in. distance from each end will satisfy the depth-restricting requirement. (See Fig. 4-135.)

Eyebolt Holes

Eyebolt holes are normally on the side of the clamping slots and should be provided on both halves opposite each other; they should be placed in areas where balanced lifting of the mold base is possible. Holes should also be tapped on surfaces perpendicular to the slots.

The forged steel eyebolts have a safe load-carrying capacity as listed:

$\frac{1}{2}$ in.	2,600 lb
$\frac{3}{4}$ in.	6,000 lb
1 in.	11,000 lb

For safety reasons, only forged steel eyebolts should be specified, preferably those with a shoulder for better stability.

Quick Mold Change

As reviewed in other chapters (Chap. 2, etc.), completely automated quick mold change (QMC) devices are being used. The cost is usually higher than that of an IMM; however, it pays for itself quickly when required. QMC, with microprocessor controls, provides cost-effective approaches to plantwide automation. Different designs are used, such as overhead or side loading/unloading platforms (Figs. 4-136 through 4-138).

The concept is best suited to processors with relatively short production runs and frequent mold changes. In such operations, the benefits of QMC are many: increased productivity, reduced inventory, increased scheduling flexibility, and more efficient processing.

Mold-changing time is wasted time. In the all-out effort to trim waste, increase productivity, and reduce inventories, quick mold changing will play an important role. Today, systematic mold changing is a novelty; in a few years, mold-changing systems, including fully automatic changers, will be much more common.

Systematic procedures to expedite mold change can take many forms: from fully automated mold conveying devices to the addition of an extra overhead crane. Regardless of how it is accomplished, QMC means reducing mold changing time to roughly 1 to 10 min and facilitating nearly instantaneous startup on the new mold.

QMC goals are achieved by standardizing the construction of molds and machine

DNA Genotek Inc. Exhibit 2029 Page 178
Spectrum Solutions LLC v. DNA Genotek Inc. IPR2023-01424

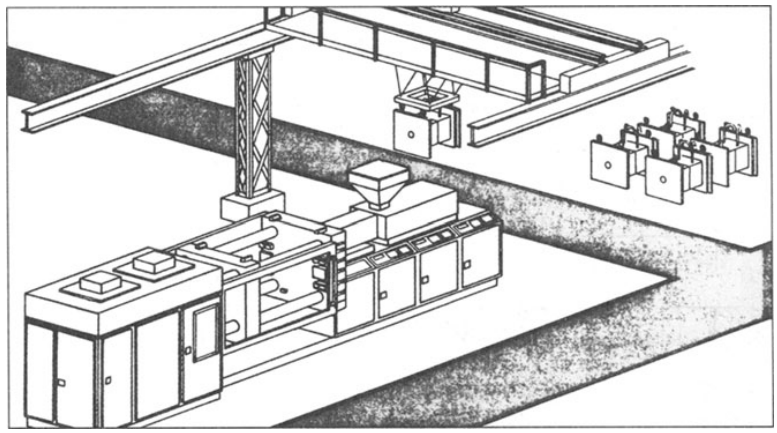


Fig. 4-136 Quick mold change system using a single robotized crane to service multiple IMMs from an inventory of 200 molds.

mounting, raising operator training levels and awareness, and increasing reliance on microprocessor-based controls.

The software stores information regarding molding cycle times and temperatures for individual molds, platen spacing for each mold change, and orchestration of the mold changing devices. Increasingly important in software and controls will be ease of programming and the ability to change programs quickly, more microprocessor storage capabilities, and reliable self-diagnostics.

Completely automated systems contain (1) mold conveyors that propel the molds in and out of position on motorized rollers, and (2) mold carriers that index on a track parallel

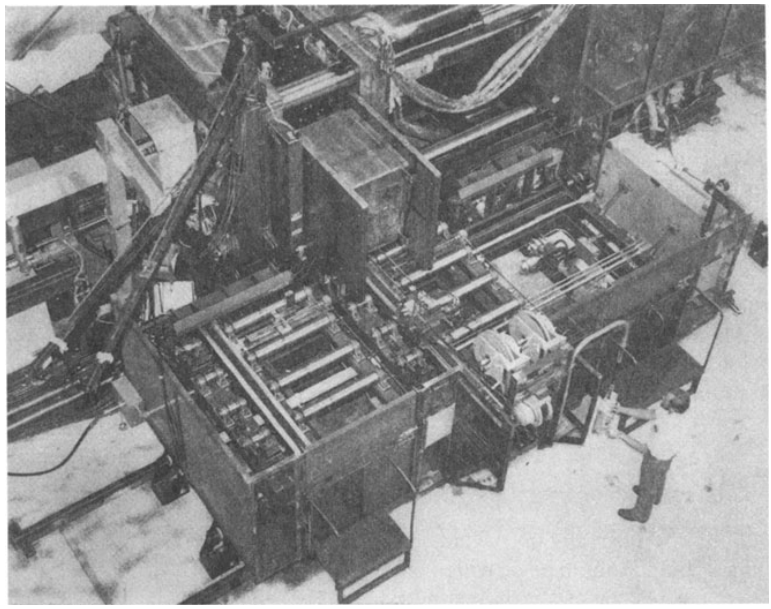


Fig. 4-137 Quick mold change device beside the IMM.

DNA Genotek Inc. Exhibit 2029 Page 179
Spectrum Solutions LLC v. DNA Genotek Inc. IPR2023-01424

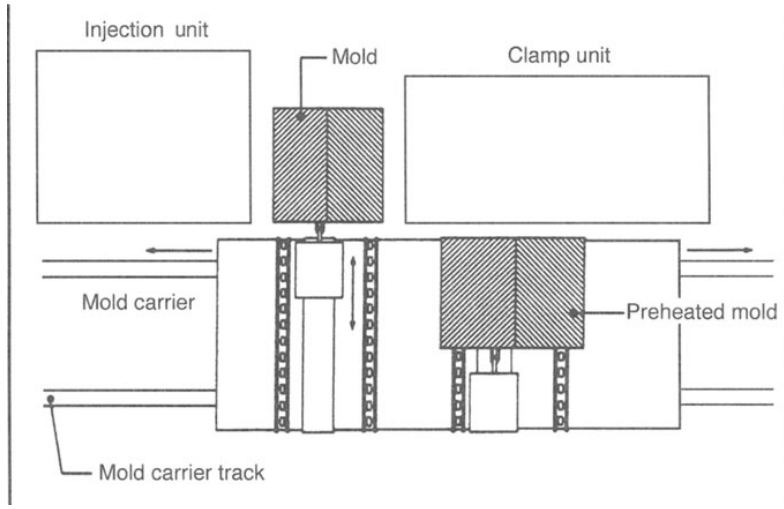


Fig. 4-138 Schematic of a fully automated quick mold change device: production interrupted for 1 to 2 min.

to the IMM to align the mold conveyors and clamp unit. They can also convey molds to and from the machines from a central mold storage area.

Basically, when the microprocessor-based control signals the end of a mold production run, the movable platen indexes to a preset position for mold removal. Automatic mold clamps are deactivated, releasing the old mold. Mold clamps or straps connect to lock together mold halves. The mold conveyor removes the old mold.

Computer control resets platens for the new mold, and the carrier table aligns the preheated new mold with the clamp unit. As the new mold is inserted by mold conveyors, automatic mold positioners center the mold within seconds. Automatic clamps are activated by the control system, and new molding-cycle information stored in the microprocessor initiates the next production cycle. Estimated downtime between old- and new-mold shot is 2 min with a highly sophisticated QMC.

Even at the design stage of a machine, consideration must be given to whether or not frequent mold change will be required. Above all, if the future customer wants JIT (just-in-time) production, it must also be possible to manufacture small numbers economically. In this situation, quick mold change devices require careful consideration.

Practical schemes that can be found for changing molds more or less automatically come in many forms, since individual requirements and boundary conditions always have a large influence. The most important points relevant to the design and overhead costs of an automatic mold change system are (1) the direction of insertion (or removal) of the mold vertically or horizontally; (2) the range of machine clamping forces to be covered by the automatic mold change system, (3) machine layout, machine accessibility, and material flow; (4) the number of molds per machine; (5) the mold-related batch size; (6) the ability of the customer to accept the product delivery and time requirement to be met, such as JIT; and (7) the off-load schedule for the IMM (time allowed for a mold change).

The current efforts at standardization of mold clamping and coupling represent an important step. They should make decisions about automatic mold change easier for fabricators. Systems that have been installed so far cost relatively little. Increasing acceptance of snap-fit rapid clamping systems should induce many injection machine manufacturers to alter platen dimensions and distances between tie-bars.

DNA Genotek Inc. Exhibit 2029 Page 180
Spectrum Solutions LLC v. DNA Genotek Inc. IPR2023-01424

The general agreement of machine design in favor of an extensive horizontal exchange system, independent of manufacturer, is not expected soon because of the far-reaching changes to IMMs that are required. Because more difficult requirements are associated with automatic mold change than the adaptation of robots, the IMM manufacturers and their trading partners are likely to meet only specialty customer requirements for the immediate future.

Mold Protection

A tool that has received all the necessary attention and care from the designer and moldmaker should be handled with extreme care so that the expanded effort is fully protected. Any protruding parts should be protected against damage in transfer. The mold surfaces, especially cavities and cores, should be covered with a protective coating against surface corrosion. The coating should be easily removable before the molding operation starts.

The protection of mold surface applies equally to the time after a run, when the mold is ready to be removed from the press and stored for the next run. In some areas where the atmosphere is highly corrosive, the mold must be protected while in the press for anticipated operation. This is especially important over a long holiday weekend of 72 h or more. Commercial coatings are available for this purpose; before they are used, however, they should be carefully evaluated for their ability to protect the area involved. Also, vacuum containers are used after molds are properly dried.

Automatic Systems

When an IMM is converted to automatic molding, its operator is dispatched to other tasks, or assigned to stay close to multiple machines. Closed loops and various devices take over the operator's two principal tasks: part removal and minor machine adjustments. When parts, are removed automatically, it is necessary to ensure automatically that they have, in fact, been removed. Thus, the need exists for mold protection: The part (or sprue or runner) that lingers in the mold or hangs up while exiting is a potential source of mold damage.

If a robot or automatic part-removal system moves in between the platens and removes the parts, there usually is no further concern about mold protection, because these devices are fitted with sensors that will signal a shutdown if the parts elude their grasp. When the parts are merely ejected and dropped, there are two general approaches: a mold-protection circuit built into the machine controls, and an assortment of add-on devices that either detect any parts that have not been ejected or record their departure.

Today's IMMs have built-in mold protection at varying levels of sophistication. It is part of the low-pressure closing portion of the cycle. If ejection has not been successfully performed, the mold cannot close completely in the low-pressure mode. Various combinations of pressure, position, and time readings react to reopen the platens. An alarm may then be sounded, or a "try again" cycle initiated, reclosing to determine if the partial close succeeded in dislodging the offender, sometimes preceded by ejection-system pulsing.

Built-in mold protection has become more effective because of the increased sophistication in hydraulics, control systems, and such discrete components as optical encoders. As would be expected, there are subtle differences between the various OEM systems.

Heavy Molds

With large molds that require large machines such as 2,000 tons and above, the need for changes in the normal operations and procedures used with smaller machines is apparent. A key difference lies in the installation and support of the large molds in the machine. Generally, each mold installation depends on the experience and skill of the personnel involved.

Injection molds have traditionally been attached to the molding machine with friction

DNA Genotek Inc. Exhibit 2029 Page 181
Spectrum Solutions LLC v. DNA Genotek Inc. IPR2023-01424

clamps. These clamps are bolted directly to the platens and, in turn, press the mold against the platen, holding it there by frictional force. As the size and weight of molds have increased in recent years, some production runs have been able to use other clamping methods, such as hydraulically actuated clamps or direct bolting of mold halves to the platens. Whatever mode of attachment is used, it must be adequate to keep the mold from falling or being pulled off the platen (179).

To determine the required number of mold clamps, it is necessary to analyze the forces that might allow a mold to fall and the components in place to resist those forces. The two major forces are gravity (vertical) and machine mold-opening force (horizontal).

The dependence on frictional forces to support molds against gravity is plagued with uncertainties, and the required number of mold clamps is influenced by numerous factors, including:

- Weight of the mold half
- Coefficient of friction, as affected by the presence of oil or lubricants on the mold and/or platens
- Clamp bolt torque levels
- Clamp bolt diameter
- Distance of clamp bolt (location in clamp slot) from mold
- Parallelism of mold clamps
- Presence or lack of (1) a recessed locating ring in the stationary mold half, and (2) ejector return rods tied to the machine ejector return on the moving mold half
- Presence or lack of additional support systems
- The margin of variability engineered into the system (the safety factor)

The most significant of the above factors in the estimation of clamp holding power are the *friction coefficient, bolt torque level, bolt diameter*, and *location of bolt in the clamp slot.* The coefficient of friction between the various clamping surfaces with a large mold clamp has been determined to be about 0.07.

The next significant parameter in the estimation of holding power is the torque applied to the clamp bolts. When torque is applied manually, a conservative estimate of bolt torque is 200 ft-lb (2,770 kg-cm) [100-lb (45.4-kg) force applied with a 2-ft (61-cm) wrench]. Measurements were made on torque applied with a $\frac{3}{4}$-in.-drive pneumatic impact wrench for up to 15 sec. Although the rated capacity was 750 ft-lb (10,390 kg-cm), the average actual torque measured was 381 ft-lb (5,280 kg-cm) with a three-sigma variation (three standard deviations from the mean) as low as 283 ft-lb (3,920 kg-cm).

Therefore, measurement of the final torque level should be tested and determined with an instrumented torque wrench. In many cases, this is cumbersome or impossible to do. If it cannot be measured, we use the conservative assumption of 280 ft-lb (3,880 kg-cm) for the specific wrench. The larger the diameter of the bolt, the less delivered force occurs at a given torque level. The design of a typical friction clamp required for the 4,000-ton machine platens has a $1\frac{1}{4}$-in. (3.2-cm) bolt diameter, which would deliver only 80% of the force of a 1-in. (2.5-cm) bolt at the same torque level. The distance of the bolt from the nose of the clamp can vary from about 2 to 6 in. (5.1 to 15.2 cm). The larger the distance, the less holding force is generated at a given torque level.

The presence of a recessed locating ring in the mold half, which mates against a projecting ring mounted in the platen in the nozzle area, can serve as an additional support of the stationary mold half against slippage.

Similarly, attaching mold ejector return rods to the machine ejector return provides additional support for the moving mold half, but this is not a universal design feature. The parallelism of the mold clamp to the platen is also a necessary requirement for proper mold support.

Accurate determination of the mold-to-platen clamping forces involves rather lengthy equations and requires test data on the friction coefficients within the system being considered. However, the process can be simplified with reasonable confidence by utilizing "typical" friction values and applying a conservative safety factor to compensate for the uncertainties. Only the horizontal clamp-opening force (vs. the shear force of the

DNA Genotek Inc. Exhibit 2029 Page 182
Spectrum Solutions LLC v. DNA Genotek Inc. IPR2023-01424

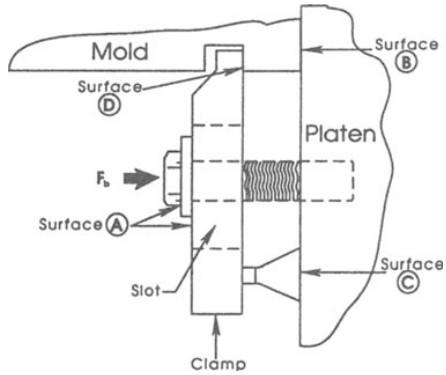


Fig. 4-139 Bottom friction clamp must prevent slippage at points A, B, and C in order to hold the mold securely.

bottom bolts) is considered, because of the uncertainties surrounding the variety of mold and clamp designs that might allow any mold slippage to exceed the clearance between the top clamps and top of the mold.

A typical bottom clamp arrangement is shown in Fig. 4-139. For the mold to slip downward, slippage must occur at surfaces *A*, *B*, and *C*. Through static analysis of this arrangement, if we assume the coefficients of friction at the three surfaces are the same, these simplified equations can be developed:

Bolt weight:

$$F_b = T/(KD)$$

where F_b = bolt force (lb)
T = bolt torque (in.-lb)
K = a geometric and friction factor relating the bolt threads and head/washer friction, and the thread design (a typical textbook value is 0.2; ours was 0.13)
D = bolt diameter (in.)

Weight capable of being supported by bottom clamps:

$$W_b = n_b(2uF_b)/F$$

where W_b = weight supported by frictional forces (pounds)
n_b = number of lower clamps
u = coefficient of friction at surfaces *A*, *B*, *C* (a typical textbook value is 0.15; ours was 0.07)
F_b = bolt force (lb)
S = safety factor

In a typical top clamp arrangement, for the mold to slip downward, slippage must occur only at surfaces *B* and *D*. If we assume that the coefficient of friction at these two surfaces is the same, these simplified equations can be developed:

Bolt force is calculated the same way as with the bottom clamp.

Weight capable of being supported by top clamp:

$$W_t = n_t(2uF_b)X/(YS)$$

where W_t = weight supported by frictional forces (lb)
n_t = number of upper clamps
u = coefficient of friction at surfaces *B*, *D* (a textbook value is 0.15; ours was 0.07)
F_b = bolt force (lb)
X = distance from clamp foot centerline to bolt centerline (in.)
Y = distance from clamp foot centerline to end of clamp nose (in.)
S = safety factor

Total weight supported by frictional forces:

$$\begin{aligned} W_T &= W_b + W_t \\ &= n_b(2uF_b)/S + n_t(2uF_b)X/(YS) \\ &= (2uF_b/S)(n_b + n_tX/Y) \end{aligned}$$

where $F_b = T/(KD)$ as noted above.

For example, in the hypothetical case of an 8,000-lb mold half using four clamps on the mold top and four clamps on the bottom, torque being applied manually, and bolts located farthest from the mold: T was estimated at 2,400 in.-lb; $D = 1.25$ in.; $n_b = 4$; $n_t = 4$; $X = 2.25$ in.; $Y = 11.75$ in. For the purpose of analysis, we will assume that $S = 1$. Two cases will be presented: (1) for typical textbook values for K and u, for 0.2 and 0.15,

DNA Genotek Inc. Exhibit 2029 Page 183
Spectrum Solutions LLC v. DNA Genotek Inc. IPR2023-01424

respectively, and (2) for measured values of K and u, 0.13 and 0.07, respectively.

Case (1):

$$F_b = (2{,}400)/[(0.2)(1.25)]$$
$$= 9{,}600\text{-lb } (4{,}358\text{-kg}) \text{ force}$$
$$W_T = (2 \times 0.15 \times 9{,}600/1) \times (4 + 4 \times 2.25/11.75)$$
$$= 13{,}726 \text{ lb } (8{,}232 \text{ kg})$$

Case (2):

$$F_b = (2{,}400)/[(0.13)(1.25)]$$
$$= 14{,}769\text{-lb } (6{,}705\text{-kg}) \text{ force}$$
$$W_T = (2 \times 0.07 \times 14{,}769/1) \times (4 + 4 \times 2.25/11.75)$$
$$= 9{,}854 \text{ lb } (4{,}474 \text{ kg})$$

Since the actual mold half weighed 8,000 lb (3,630 kg), the safety factor was 1.7 using textbook values and 1.2 using the measured values. This obviously underlines the need for using large safety factors when depending on frictional holding forces.

If the mold half is directly bolted to the machine platen, the vertical support mechanism is no longer totally that of friction, but includes the shear resistance of the bolts supporting the weight. This is usually true *only* for the bottom bolts and/or side bolts, and only if their vertical clearance to the top of the slots does not exceed that between the top bolts and top of the mold. In other words, the lower supporting bolts still cannot allow the top of the mold to slip below the top bolts.

In the above case, the four bottom $1\frac{1}{4}$-in.-(3.2-cm)-diameter bolts could support the weight of 13 tons with a safety factor of 6. It is obvious that this is strongly preferred over using friction clamps. It is best to design or adapt molds for direct bolting whenever possible.

Selection of the proper safety factor is very important and will depend on the specific molding operation. If an operator must routinely place part of his or her body between the mold halves, this factor must be high, because of the uncertainties of frictional forces. A popular safety factor for designing material-handling equipment, such as hoists or cranes, is 5.

As mold sizes increase in each dimension, their weights increase exponentially. Depending on the combination of mold geometry and mold clamp design, the number of required clamps for a mold half can soon reach values of 15, 20, or even 40. This, of course, can be impractical and often impossible to achieve. Alternative methods of mold support are then required. In addition, with large machines where operators will be required to enter the space between mold halves for part removal or other tasks, it is best that the mold not be supported by friction clamps alone. Thus, molds should be supported by two additional systems, namely, a jack block and an overhead support system.

The jack block support system entails bolting two $6\frac{1}{2} \times 10$-in. (16.5 × 25.4-cm) steel blocks to each platen with $1\frac{1}{4}$-in. (3.2-cm) bolts at a level below the bottom of the mold. Each block has a vertically adjustable 2-in.-(5.1-cm)-diameter screw that can be raised up to 20 in. (50.8 cm) to contact the clamped or bolted mold half. This positive vertical support system effectively prevents a mold half weighing up to 15 tons from slipping.

The overhead support system involves bolting two 5-in.-(12.7-cm)-thick overhead rigging arms with two $1\frac{1}{2}$-in.-(3.8-cm)-diameter bolts each to the top of each platen. The arms are slotted and can swivel so that a chain assembly can be attached almost vertically from an eyebolt in each arm to eyebolts in each mold half. This system is designed to suspend a mold half weighing up to 15 tons and will prevent any significant vertical movement should the mold clamps and jack blocks be insufficient support for any reason.

Large injection machines can have substantial designed-in maximum breakaway or mold-opening forces. As an example, a 4,000-ton machine has an available maximum breakaway force of 160 tons. Although normally such maximum forces are not reached, it is, nonetheless, possible to do so in trying to overcome the various forces developed between the part and core of deep-draw parts. If an insufficient number of clamps or bolts is used, the mold half can be pulled off the

DNA Genotek Inc. Exhibit 2029 Page 184
Spectrum Solutions LLC v. DNA Genotek Inc. IPR2023-01424

machine platen. Depending on the mold design, it could fall to the floor. Although this does not pose the same risk to personnel, since the operator(s) would be outside the machine at this time, it is of concern for the cost and downtime to repair the mold.

For the industry-standard friction clamp with a $1\frac{1}{4}$-in. (3.2-cm) bolt, calculations show that for a safety factor of 2 in the worst-case situation (bolt located furthest from the clamp nose), some 30 clamps per mold half would be required to withstand the full 160-ton opening force of the 4,000-ton machine. This is impractical and, in most cases, impossible to implement. A solution is to work with the machine control software so that a maximum opening force of only 80 tons can be achieved. Therefore, the minimum number of mold clamps per mold half is specified as 15. Should additional opening force be necessary, a new mold clamp design would be required.

If the mold half is direct-bolted to the machine platen, a significant improvement occurs. In this case, only 10 $1\frac{1}{4}$-in. bolts will withstand the full 160-ton force with a safety factor of 5.

The above standards for mold mounting are supplemented by a series of operating procedures designed to ensure safe and proper handling of large injection molds prior to, during, and after being used in the molding operation. These are:

1. *Mold acceptance.* A checklist that includes minimum design standards that must be met or adapted prior to installation of any mold.

2. *Premold installation.* A checklist aimed at preparing the machine to receive the mold.

3. *Mold transportation.* A procedure for moving the mold by overhead crane.

4. *Mold clamping.* A detailed method of determining the minimum number of mold clamps per mold half, minimum torque requirements, and methods of applying the latter.

5. *Jack block support.* Details of initial installation and adapting to any mold installation.

6. *Overhead support.* Details of initial installation and adapting to any mold installation.

7. *Mold installation.* Step-by-step procedures for installing molds, which refer to most of the above procedures.

8. *Mold removal.* Step-by-step procedures for removing molds, which refer to the above procedures.

Preengineered Molds

Within the industry, some manufacturers have developed mold standardization programs. In choosing the number of cavities per mold, consideration should be given to the standard molds available. There are benefits to mold standardization; for example, (1) high-quality manufacturing techniques result in consistent quality and reduced mold cost; (2) there is improved delivery time, with only the core and cavity having to be machined, as other components can be inventoried by the manufacturer of the preengineered parts; and (3) mold performance can be closely predicted, based on the past experience of the manufacturer or molder. Thus, you can obtain the required dimensional accuracy, close tolerance, high-quality steels, and interchangeability. (See the section on mold components above and also the Terminology section at the end of this chapter.)

Table 4-28 provides information on some of the manufacturers that produce preengineered mold components (Fig. 4-140). Some of these companies specialize in specific components, such as Mold-Masters Ltd—hot-runner systems; Incoe—special gate controls to automatically control resin flow into the cavity(s); Master Unit Die Products (MUD)—quick-change cavity mold; Logic Devices—mold-venting devices; and 3M—custom-molded cavities.

Certain companies, such as Husky, have extensive preengineered mold capability, but can also package a mold around a complete IMM with all types of parts-handling equipment, such as robots. (See Chap. 10 on parts-handling systems.) Husky is a major world

DNA Genotek Inc. Exhibit 2029 Page 185
Spectrum Solutions LLC v. DNA Genotek Inc. IPR2023-01424

Table 4-28 Manufacturers of preengineered mold bases and components

ABA Tool & Die Co., Manchester, CT
Alliance Mold Co., Inc., Rochester, NY
Bermer Tool & Die Co., Southbridge, MA
Chromalloy Div., Sintercast, West Nyack, NY
Columbia Engineering, Red Lion, PA
D-M-E, Milicron, Madison Heights, MI
Erico Products, Solon, OH
Ethyl/VCA Marland, Inc., Pittsfield, MA
Fast Heat Element Mfg., Elmhurst, IL
G-W/Carborundum, Bethel, VT
Husky Injection Molding Systems Ltd., Bolton, Ontario
IMS Co., Cleveland, OH
Incoe Corp., Troy, MI
Industrial Heater, New York, NY
ITT-Vulcan Electric, Kezar Falls, ME
Kona Corp., Gloucester, MA
Logic Devices, Bethel, CT
3M Custom Molded Products, St. Paul, MN
Master Unit Die Products, Inc., Greenville, MI
Mold-Base Industries, Inc., Harrisburg, PA
Mold Masters Ltd., Toronto, Canada
National Tool & Mfg., Kenilworth, NJ
Newark Die Co., Springfield, NJ
Parker-Hannifin, Quick-Coupling Div., Elyria, OH
Sno-Trik, Solon, OH
Stilson Div., Stocker & Yale, Roseville, MI
Value Molding Corp., Loveland, CO

manufacturer of IMMs, specializing in producing the complete injection molding line (mold, machine, parts-handling, etc.), with components that operate at very fast rates producing quality products. D-M-E Company provides the industry with almost all types of preengineered mold bases and components. Some D-M-E Milicron products will be reviewed later in this chapter to give examples of what is available. Manufacturers each have their own booklets or manuals describing their products and how *one* best can simplify and operate their molds with tight control.

Preengineered mold components provide the same important economic and technical advantages as standard mold bases: dimensional accuracy, interchangeability, availability, etc. These components can be divided into various categories, including basic mold components, alignment and registry components, heating and cooling items, and specialty components (Fig. 4-140).

Basic mold components include items such as ejector pins and sleeves, used to eject plastics parts from the mold; leader pins and bushings, used to maintain mold alignment when the mold is removed from the press; sprue bushings, installed in the mold to accept the plastic melt from the molding machine

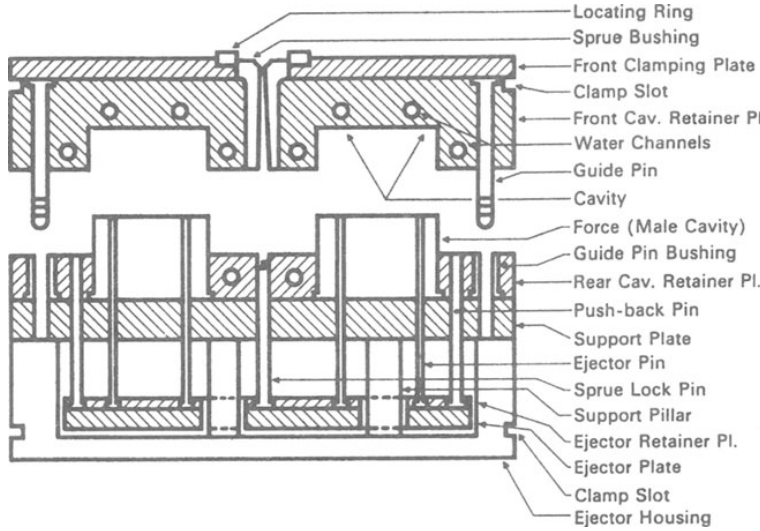


Fig. 4-140 Preengineered standard mold.

nozzle; locating rings, installed in the stationary half of the mold to locate the machine nozzle with the sprue bushing; and support pillars, used to increase the capacity of the mold to support the projected area of the cavities, runner, and sprue.

Alignment and registry components include tabular dowels, used in mold base assemblies to accurately align the B plate, support plate, and ejector housing; and round and rectangular tapered interlocks, used when very accurate registration of mold halves, mold plates, or individual cavities and cores is required.

Preengineered heating items include components such as helical tabular heaters, used for heating IMM nozzles; band heaters, designed for heating mold plates, probes, injection cylinders, and nozzles; and thermocouples, used to monitor temperatures.

Cooling components include a variety of brass items for controlling water temperature and flow within the mold. These components take various forms, such as bubbler tubes; cascade water junctions; plug baffles; pressure plugs; diverting plugs and rods; and Jiffy-Matic connectors (plug-and-socket components used to provide quick connect/disconnect of water lines).

Standardized Mold Base Assemblies

Of the thousands of standardized mold base assemblies offered by D-M-E, the most popular is the A series (Fig. 4-141). This most frequently used assembly is available in different sizes (Fig. 4-142) from $7\frac{7}{8} \times 7\frac{7}{8}$ in. to $23\frac{3}{4} \times 35\frac{1}{2}$ in. It has been preengineered in cooperation with experienced mold designers to accommodate the widest variety of injection molding applications. The A series mold base suits most plastics part requirements, simplifies mold design, increases moldmaking productivity, and gives the molder the most economical, high-performance mold construction. It permits through-pocket machining for cavity and core inserts in the cavity retainer plates, reducing mold machining time and costs.

The B series mold bases are a modification of the A series design made with the same steels, interchangeable component parts, and precision manufacturing (Fig. 4-141). The fundamental difference is its two-plate design vs. the four-plate assembly provided in the A series. The B series mold base uses the cavity retainer plates for a dual purpose, eliminating the need for a separate top clamping plate and support plate. Multiple-cavity molds designed as part of the B series require that the cavities and cores be inserts into blind pockets machined into the cavity retainer plates. The B series is sometimes specified for single-cavity plastics molds where the cavity and core are machined directly in the cavity retainer plate, or overall mold height is critical.

For applications requiring stripper plates for part ejection, the D-M-E X series mold bases can be used (Fig. 4-141). Two versions of this mold base are available: the six-plate series, with a support plate, and the five-plate series, without a support plate.

Another variation of the standard A series mold base is called the AX series (Fig. 4-141). The AX base is basically an A series type with a floating plate (X plate) added between the cavity plates. This type of assembly is used when it is desirable to have the floating plate remain with the upper half of the assembly, for example, when runners are top-mounted. Another group of mold bases is designated the T series (Fig. 4-141). These bases are used for top-runner molds that require two floating plates (X-1, the runner stripper plate, and X-2, the cavity plate) to remain with the upper or stationary half of the assembly.

In addition to the standard mold bases described above, there are designed and engineered custom mold bases available for specific molding machines. These custom-designed bases include a variety of configurations, including the shuttle type and universal type with adapter plates.

Mold base component parts, such as cavity retainer sets, mold plates, die blocks, spacer blocks, and ejector housing, are available for those cases in which standard assemblies cannot be used (Fig. 4-143). The broad line of standard mold base component parts available from different companies permits the

DNA Genotek Inc. Exhibit 2029 Page 187
Spectrum Solutions LLC v. DNA Genotek Inc. IPR2023-01424

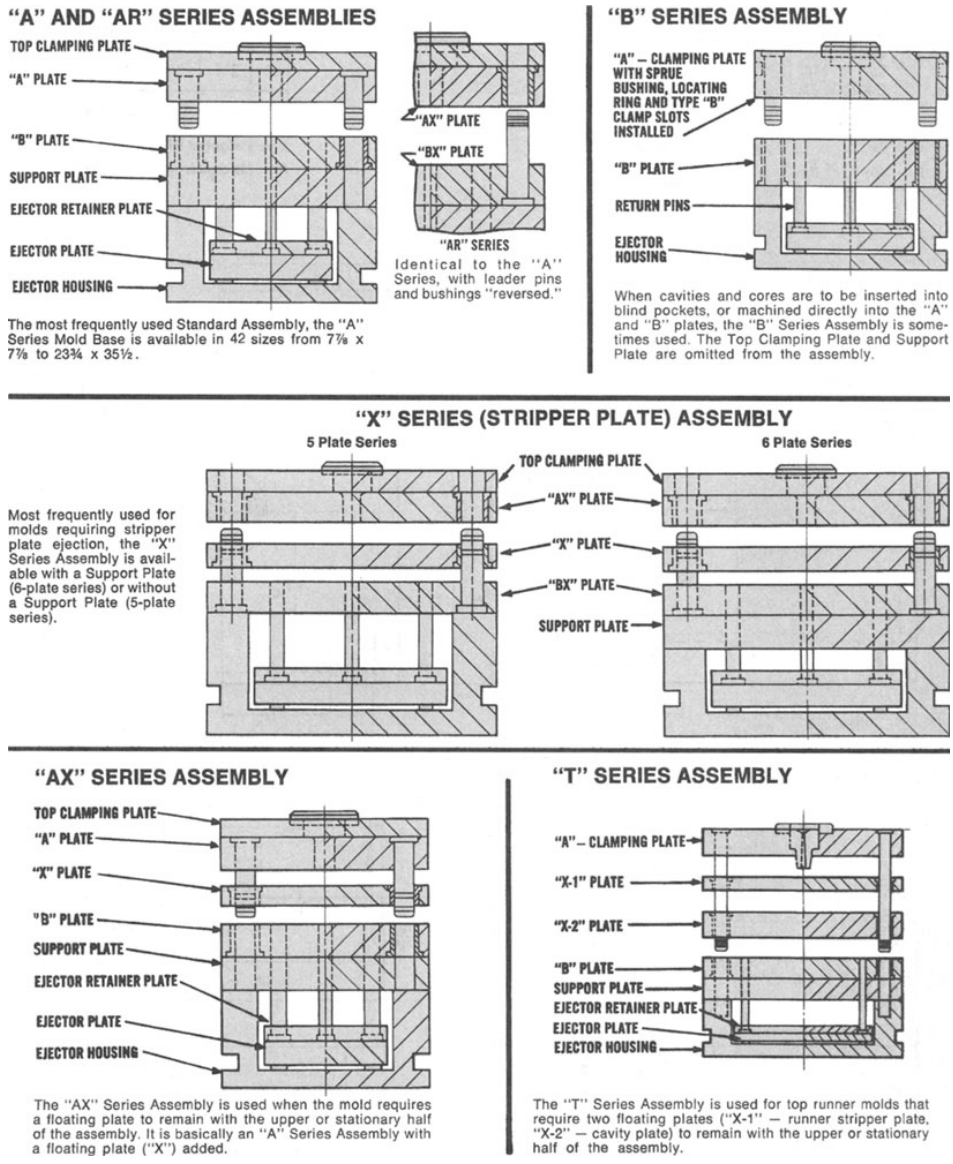


Fig. 4-141 Preengineered standard DME mold bases.

design and construction of custom mold assemblies, while retaining the important advantages of interchangeability. Plates and components for large mold assemblies are also available, providing the benefits of standardization for large tooling applications as well.

Specialty Mold Components

Specialty components are those that have been engineered to improve the performance of particular mold functions. These functions can be as straightforward as returning the ejector assembly early in time or as

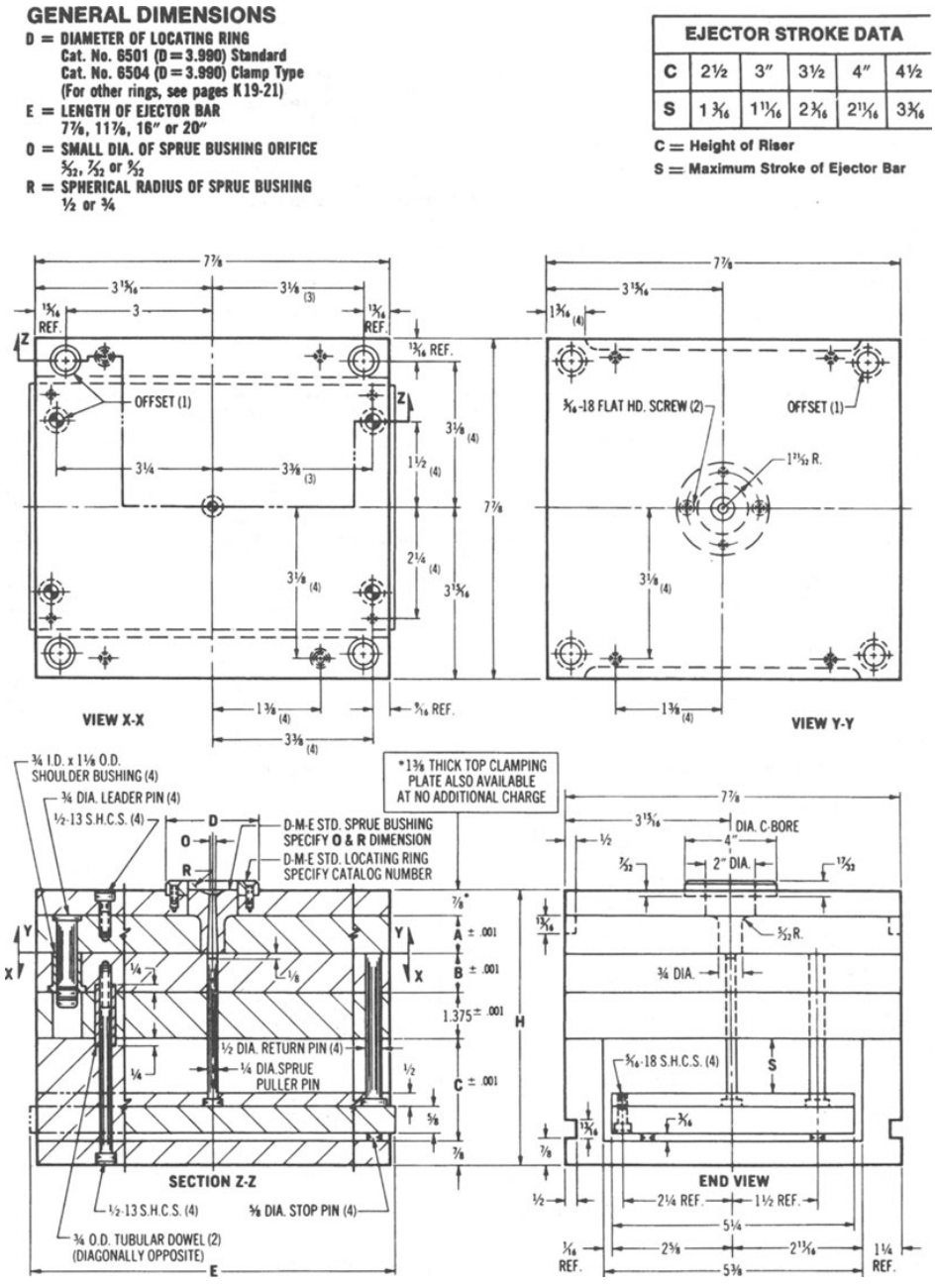
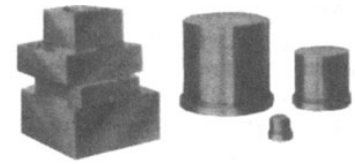


Fig. 4-142 Examples of DME standard A series mold bases.

Cavity Insert Blocks and Rounds

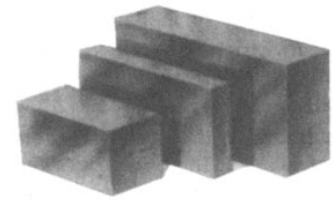
Cavity Insert Blocks are stocked in over 90 standard sizes, from 3″ x 3″ to 6″ x 8″; ⅞ to 4⅞ thick. They are available in your choice of D-M-E No. 3 (P-20 type) or No. 5 (H-13 type) steel. The more popular sizes are also available in D-M-E No. 6 (T-420 type) stainless steel.

Cavity Insert Rounds are available in 41 standard sizes, from one to four inches in diameter, ⅞ to 3⅞ long. They are stocked in both D-M-E No. 3 and No. 5 cavity steels.



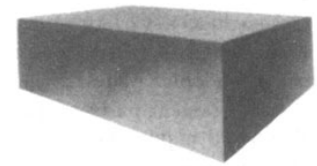
Die Blocks and Plates — No. 5 Steel

Available in over 300 standard sizes from 7⅞ x 7⅞ to 23¾ x 35½; 1⅜ to 11⅞ thick (depending on length and width). They are supplied in milled condition, with approximately .060″ stock allowance.



Extra Thick Mold and Die Blocks

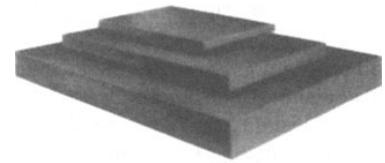
Available in D-M-E No. 1, No. 2 or No. 3 Steel, in over 100 standard sizes, from 14⅞ x 17⅞ to 23¾ x 35½; 6⅞ to 11⅞ thick. These blocks are supplied in milled condition with approximately .060″ stock allowance.



Mold Plates and Plate Items

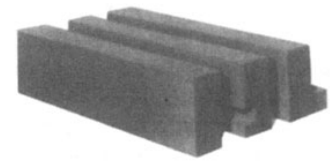
Mold Plates are available in over 400 standard sizes, from 6″ x 7″ to 23¾ x 35½ in D-M-E No. 1, No. 2 or No. 3 Steel. They are finish ground top and bottom to a thickness tolerance of plus or minus .001″ with all edges finished square and parallel.

A wide range of other mold plate items available as standard include:



Spacer Blocks

Plain, Slotted and Angle Spacers are all made from D-M-E No. 1 Steel. Riser height (C dimension) is finish ground to plus or minus .001″.



Ejector Housings

Rigid one-piece construction is made from D-M-E No. 1 Steel. Available in over 150 standard sizes, corresponding to D-M-E Standard "A" Series Mold Bases. The riser height (C dimension) is finish ground to plus or minus .001″.

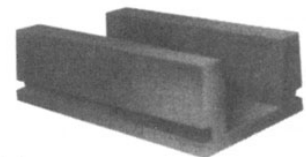


Fig. 4-143 Mold base DME component parts.

sophisticated as a runnerless molding system. The important point is that all these devices have been standardized for installation in a variety of molding applications, and as a result, they do not have to be designed and built from scratch by individual mold designers and moldmakers.

Accelerated ejectors use a rack-and-pinion mechanism to provide up to $\frac{5}{8}$ in. (1.59 cm) of additional ejector stroke. Their simple, linear movement can be used to increase the speed and stroke of ejector pins, ejector sleeves, or entire ejector assemblies. The flanges and rounded corners on these units facilitate installation within the ejector assembly. The rectangular cross section of the racks prevents them from rotating. Included with each unit is a bumper stud that assures the positive return of the racks when the ejector assembly is fully returned. Accelerated ejectors are available in two sizes (small or regular) and two types (pin or bumper). The pin-type units are used for individual ejector pin acceleration (one unit per pin). Bumper-type units are used for accelerating the entire upper ejector assembly in a dual-ejector assembly mold (a minimum of four units are normally used in this application).

Accelerated knockouts are simple in design, using a pivot-type motion for accelerated ejection; the mechanical advantage is 1 : 1. The simplicity of design permits accelerated knockouts to be either inserted into the ejector plate or top-mounted, depending on the space available for the ejector movement.

The Jiffy Latch-Lok (D-M-E) provides new freedom in design to float plates mechanically. There is no need for electric switches, pneumatic controls, or timing devices with delicate adjustments. The action of the Latch-Lok is positive. Once properly installed, it eliminates the possibility of smashing the mold, because there are no adjustments that can change or connections that can be accidentally knocked off. The Jiffy Latch-Lok is available in sizes for regular or heavy-duty operation. It also comes in regular and 90° (right-angle) designs to provide maximum installation flexibility.

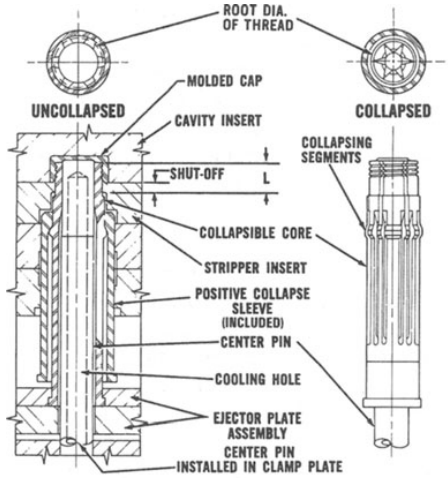
The slide retainer provides a compact and economical means of slide retention that makes obsolete the cumbersome external spring or hydraulic methods. Its simple and positive operation makes it equally suitable for new tooling design or retrofitting existing molds. Available in three sizes with increasing weight-holding capacities, the slide retainers can be used individually or in multiples for larger or heavier slides. Generally mounted behind and below the slide, the slide retainer is a compact unit that is entirely contained within the mold. Interference with machine tie-bars or safety gates is not a problem. It can even be installed completely underneath the slide if space is limited.

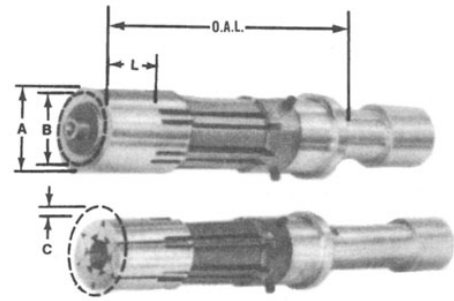
As the mold opens, the dowel pin installed in the slide positively locks into the retainer until disengaged by the mold's closing action. The small spring placed crosswise in the retainer maintains the gripping force required to keep the dowel pin in the socket when the mold is open. The slide retainer is designed with a generous lead-in at the socket opening so the dowel pin will enter the socket even if there is a slight misalignment between the retainer and pin. This investment-cast unit includes an integral protective cover over the spring, preventing foreign objects from interfering with the spring's action.

The pneumatically controlled and operated Jiffy-Jector (D-M-E) is a compact, powerful device for positively ejecting runners from three-plate molds. It moves the runner system away from the X-1 plate and then, with a short positive stroke and blast of compressed air, ejects the runner system down and out of the mold, thereby ending hangup problems. The basic requirement for proper operation of the Jiffy-Jector is that the runner system has a clear, unobstructed path out of the mold. It is adaptable to most three-plate molds and can be designed into new molds or retrofitted to existing molds.

The collapsible core is a major improvement for molding plastic parts requiring certain complex parts (Fig. 4-144). It provides a means to mold internal threads, undercuts (as in tamper-proof bottle caps, etc.), protrusions, cutouts, etc. There are more than a dozen patents approved on the design of collapsible cores. Many of these designs will never see the market due to the complexity of

DNA Genotek Inc. Exhibit 2029 Page 191
Spectrum Solutions LLC v. DNA Genotek Inc. IPR2023-01424





CATALOG NUMBER	A Max. O.D. of Thread or Configuration		B Min. I.D. of Thread or Configuration		Center Pin Dia. (At Top of Collapsible Core)		L Max Molded Length (Incl. Mold Shut-Off				C Collapse per Side at Tip of Core††				O.A.L. Overall Length of Collapsible Core (Only)	
	inch	mm	inch	mm	inch	mm	inch	mm	*inch	*mm	inch	mm	*inch	*mm	inch	mm
CC-200-PC	1.270	32.25	.910	23.11	.785	19.93	.975	24.76	1.150	29.21	.043	1.09	.048	1.21	7.315	185.80
†CC-250-PC	1.270	32.25	.910	23.11	.785	19.93	.975	24.76	1.150	29.21	.043	1.09	.048	1.21	5.440	138.17
CC-202-PC	1.390	35.30	1.010	25.65	.885	22.47	.975	24.76	1.150	29.21	.055	1.39	.064	1.62	7.315	185.80
†CC-252-PC	1.390	35.30	1.010	25.65	.885	22.47	.975	24.76	1.150	29.21	.055	1.39	.064	1.62	5.440	138.17
CC-302-PC	1.740	44.19	1.270	32.25	1.105	28.06	1.225	31.11	1.400	35.56	.068	1.72	.083	2.10	7.315	185.80
†CC-352-PC	1.740	44.19	1.270	32.25	1.105	28.06	1.225	31.11	1.400	35.56	.068	1.72	.083	2.10	6.065	154.05
CC-402-PC	2.182	55.42	1.593	40.46	1.388	35.25	1.535	38.98	1.700	43.18	.090	2.28	.103	2.61	7.815	198.50
CC-502-PC	2.800	71.12	2.060	52.32	1.750	44.45	1.750	44.45	1.900	48.26	.115	2.92	.125	3.17	9.625	244.47
CC-602-PC	3.535	89.78	2.610	66.29	2.175	55.24	2.125	53.97	2.400	60.96	.140	3.55	.148	3.75	11.250	285.75

Fig. 4-144 Standard DME collapsible core.

the parts or high tooling cost. However, there are two designs in production in the United States, commonly known as a standard collapsible core and collapsible minicore. The standard type, more often called a collapsible core, is the oldest and most popular, dating back to the 1950s. It is designed to mold circular parts with 360 undercuts. Its assembly consists of three parts: a center pin, collapsible core, and sleeve. The center pin is a precision-ground shaft with a taper on one end and flange at the other. It is of D-6 tool steel hardened to 60 to 62 Rockwell C. The collapsible core is basically a hollow cylinder with 12 matching slots parallel to the cylinder axis changing part of the cylinder into matching segments. These vertical segments are the flexing segments that form the undercut. It is made of 0 to 1 tool steel that is hardened to 56 to 58 C. The collapsing sleeve is made of 52100 steel, hardened to 50 to 54 C. The center pin expands the flexing segments of the core and provides cooling of the molding length. The collapsible core forms the undercut with the expanded flexing segments and releases the part for ejection with segments in a collapsed position. The sleeve functions as a backup unit to collapse the core segments if segments fail to collapse on their own.

The collapsible minicore was designed for a less than 1-in. (2.54-cm) closure. It has a center pin with three narrow, noncollapsing segments, a core body with three wide flexing segments attached to a common base, and a positive collapsible sleeve. The center pin's function is to expand the collapsing segments of the core and provide cooling to the core segments. The core's function is to form

the undercut with the expanded flexing segments and to release the part for ejection with the segments in the collapsed position. The sleeve functions as a backup to collapse the core segments if segments fail to collapse on their own.

Collapsible and Expandable Core Molds

There are both collapsible and expandable core systems (Collapsible Cores; see also Specialty Mold Components). The Roehr Tool Corp. (Hudson, MA) patented expandable cavity was designed to mold external details. In certain mold applications it eliminates slides, external unscrewing thread devices, etc. This device provides another means to simplify molding external threads, grooves, undercuts, or any other type of surface impression. Instead of collapsing radially inwards (as in their patented collapsible core design), the core's metal segments flower outwards (away from the axis) (Fig. 4-145) (173).

Expandable core systems can be used to provide mold design flexibility and processing advantages, particularly when molding certain complex parts. The expandable cavity was basically designed to mold external details. In certain mold applications it provides closer cavity-to-cavity locations, eliminates space consuming slides, external unscrewing thread devices, etc. This component provides another means to simplify molding external threads, grooves, undercuts, or any other type of surface impression.

The resulting reduced mold construction benefits the injection molding process in various ways that range from mold and product cost reductions to improved product performance. A major advantage is the reduction in molding cycle time, arising because the heat-transfer mold space is reduced and its behavior easier to predict. When compared with a similar conventional-size mold, the expandable device permits incorporating more cavities.

Roehr Tool Corp. has designed and manufactured special tools for precision plastic injection molding since 1964, when it introduced the collapsible core device that is used worldwide with marketing by D-M-E, part

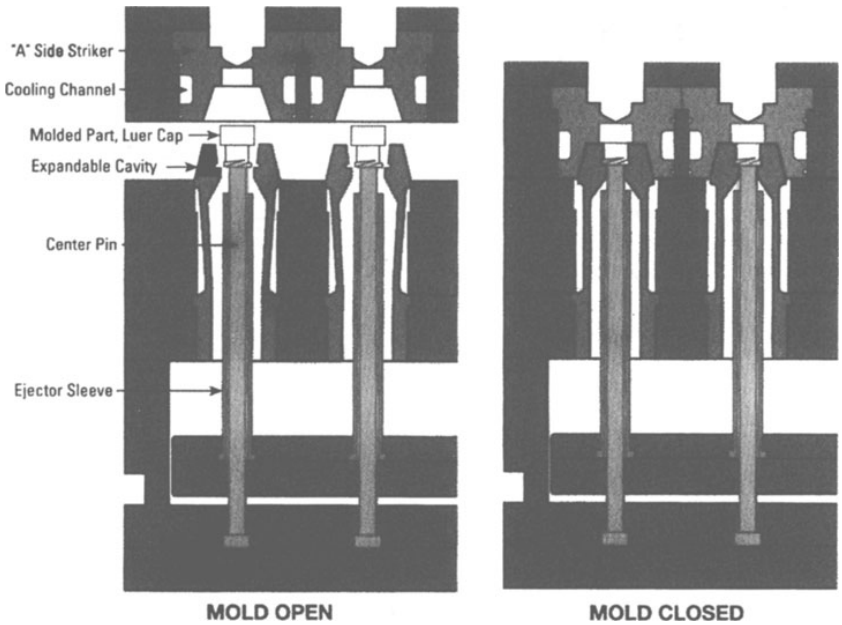


Fig. 4-145 Expandable core device in the open and closed positions.

of the Milacron Plastics Technology Group. Roehr introduced the expandable core device during the early 1980s.

Prototyping

Overview

In the past the traditional prototype was one or two engineering models of a product fabricated, usually at high cost, using standard machining techniques and equipment. In order to machine an engineering prototype, it has often been necessary to strike a compromise between the desired production material and that material's machining characteristics. The result is, at any rate, an exact model of the part that can be used to evaluate cosmetic appeal and potential fit problems. However, owing to its high cost and possible material compromise, it is not usually suited to any type of destructive property testing. Except in the simplest cases, parts machined from bar or block stock do not give the same test results as parts produced in a prototype or production mold. Also, parts produced in prototype tooling can reveal many potential molding problems before the fabrication of a production mold—problems that would never show up in a machined model.

Prototype molding provides a powerful and cost-effective tool for a designer to use when questions about a new product or potential new material arise. Questions about a part or material that arise early in the design process can be answered most definitely and cost-effectively in a prototype mold. Although the cost of a prototype mold insert set is a function of each part's individual design and requirements, it usually will run between 10 and 30% of the cost of the production mold. Another way to look at the cost of prototype tooling is to compare the information and data that it will provide with the information and data that a top-notch design staff can provide for the same cost.

Questions that require good hard answers do not always lend themselves to traditional analytical engineering solutions: questions concerning cosmetic qualities, such as finish, sink marks, witness lines from parting planes or slides, ejector pin marks, knit or weld lines, and different styles of texturing; questions concerning the moldability of a part, such as flow-through thin sections, the location of gates and vents, flow into bosses or around pins, the location of the parting plane, and potential ejection problems; and questions concerning product quality and reliability, such as shrinkage, mechanical strength of bosses and knit or weld lines, pullout resistance of molded-in inserts, electrical properties, and component fit or mating subassemblies.

The majority of the data provided from parts molded in a prototype mold can be obtained in no other more reliable or cost-effective manner. Although it is possible by using specialized computer programs to predict a materials flow path, or the location of gates, vents, knit or weld lines, or the effects of parting-plane location, with a good degree of accuracy, these programs have limitations. The highly complex parts that would benefit most by the use of these programs now overwhelm most of them; on simpler parts, the programs are not cost-effective vs. a prototype mold for the data provided.

Any part data provided from prototype tooling can also be obtained from parts produced in a production tool, but this is not a cost-effective route to follow.

Stereolithography

This is a process of creating three-dimensional plastic parts from CAD/CAM/CAE data combining four technologies: laser, optical scanning, chemistry, and software. The net effect is that complex models can be made in hours without tooling. The process takes CAD data and automatically produces a hard plastic model in a matter of hours instead of days, weeks, or longer. The model is three-dimensional and includes any design features that can be created, defined, and stored by most CAD systems in use. The basic concept starts with a design; the part design created on the CAD system is downloaded to the

DNA Genotek Inc. Exhibit 2029 Page 194
Spectrum Solutions LLC v. DNA Genotek Inc. IPR2023-01424

sterolithography apparatus with its control unit (384, 393). The control unit then directs a fine laser beam onto the surface of liquid photocurable plastic. An elevator table in the plastic vat rests just below the surface. When the 0.015-in.-(0.4-mm)-diameter laser beam hits the liquid plastic surface, it solidifies a layer 0.005 to 0.030 in. (0.1 to 0.8 mm) thick at the point of impingement.

After a part slice at one depth has been made by scanning the laser beam back and forth in the shape of the model to be developed, the elevator platform on which the model is being constructed drops by a programmed amount. Another layer or "slice" is then created on top of the first in the same manner. The process continues until the complete model has been constructed. Thus, the pattern is built from the bottom up. After laser processing, the model is then raised above the liquid level by the elevator table. The part is stripped from the table and taken to a special oven for final curing. The main advantage of this process is the speed at which a computer-generated design may be turned into a three-dimensional model that may be held, viewed, studied, and compared before a commitment to steps leading to production.

Rapid Tooling

As a result of ever-increasing advances in product design and the shortening of this process due to technology and market pressure, prototyping houses, mold makers, and toolrooms have experienced a mounting urgency to shorten lead times. Various rapid tooling (RT) program methods have been successful in offering fast toolings.

They are 3-D models suitable for use in the preliminary evaluation of form, design, performance, and material processing of molds, products, etc. When properly used, an automatic RT system can accelerate product development and improve product quality and time to the market. RT is the forming of 3-D parts (to date principally for injection molding) from the design concept to production, using computer-controlled laser beams to produce layers of the final part, whether simple or complex. Models can be made of plastics (cast epoxy, copper–nylon, etc.), metals [steel (including sprayed steel), hard alloys, copper-based alloys, powdered metals, etc.], or other materials (MIT's starch and sugar, etc.). With powder-metal molds, such models can be used as inserts in a mold ready to produce prototype parts. RP processes work from models to quickly generate tooling suitable for production of up to millions of parts (1, 7, 167, 260, 384, 414, 521, 524, 564).

Different CAD and advanced machining techniques provide faster manufacture of precision molds. There are various stereolithography systems that produce 3-D rapid prototypes in plastic, using a moving laser beam. Another example is the MIT (Massachusetts Institute of Technology) three-dimensional printing (3DP) process in which a 3-D metal mold (die, etc.) is created layer by layer using powdered metal (300- or 400-series stainless steel, tool steel, bronze, nickel alloys, titanium, etc.). Each layer is inkjet-printed with a plastic binder. The print head generates and deposits micron-sized droplets of a proprietary water-based plastic that binds the powder together. Once the layup is completed, the part is removed and placed in a sintering oven. It goes through three cycles where the plastic is burned off, the metal powder is sintered together, and the part is solidified by infiltrating with another material to fill the voids, such as a lower-melting-point metal or a plastic (epoxy, etc.). The total time is 50 h. The shape is accurate within 0.005 in. (0.0127 cm) plus 0.002 in./in. (cm/cm) and may be acceptable for prototyping. The tool can be machined to tighter tolerances and polishing. This process permits creation of any type of internal voids such as cooling lines that conform to the part shape.

The first factor in the RT is data. If you do not have all the data and the customer is still thinking about adding a rib, taking away an undercut, etc., you are not going to have a chance. It has been discovered you really

DNA Genotek Inc. Exhibit 2029 Page 195
Spectrum Solutions LLC v. DNA Genotek Inc. IPR2023-01424

need to stress to the client the importance of making all of their styling and structural changes before they go to tooling. Where a client is not in a position to make final design commitments, a RT can still be made, though then it is an additional expense. In that case the client can use the preliminary tool for visual observations to help in making final design decisions (115).

Buying Molds

Introduction

Mold prices and delivery deadlines are always the critical points in production planning and sales negotiation. The price of the mold has a major effect on the unit price of the part to be produced. Other factors having a decisive effect on successful sales are the exact calculation and rapid launching of new products onto the market. These requirements put the moldmaker under constant pressure (7).

The moldmaker is forced to limit intensive activities to the essential, that is, to the machining of cores and cavities. Everything else must be achieved in the simplest manner possible. The most economical way is to use standard elements.

Even during the stages of mold calculation and planning, standard elements at fixed prices should be considered. Today, standard elements are used to a large extent as bases for molds and dies, as well as for connections to machines and other equipment and special purposes. The use of standard elements helps to keep one within the bounds of production capacities, minimize calculation and production risks, and simplify the procurement of spare parts. The possibility of buying standardized elements on short notice considerably reduces stock keeping and shutdown times of production facilities.

Moldmaking today is moving toward more specialization. Various machine operators at milling, lathe, EDM, or jig grinding machines manufacture parts of the finished mold separately. At the final stage, all these parts are brought together and assembled by the moldmaker. The latter has the overall responsibility for the functioning of the mold, which is also a result of teamwork. Standard elements are designed exactly for this type of manufacturing process.

In principle, almost all molds consist of the same basic elements: mold plates for the inserts, intermediate plates for supporting the cores and inserts in the mold plate on the ejector side, risers to limit the working distance of the ejector plates, and clamping plates to clamp the mold to the machine.

Studies have shown that the total hours required for mold production involve 25% mold construction, 20% additional work on the mold, and 55% contour of parts. At least 25% of the required capacity can be saved by using standard elements. In addition, special machining requirements can also be handled by the manufacturer of standard elements. In total, this adds up to some 40% of the capacity required for the production of a mold.

A moldmaking shop that replaces rough machining capacity by specialized machining capacity for the production of contour parts utilizes manpower and machines more efficiently. Calculations can be made with lower hourly rates, since the rate of utilization is better.

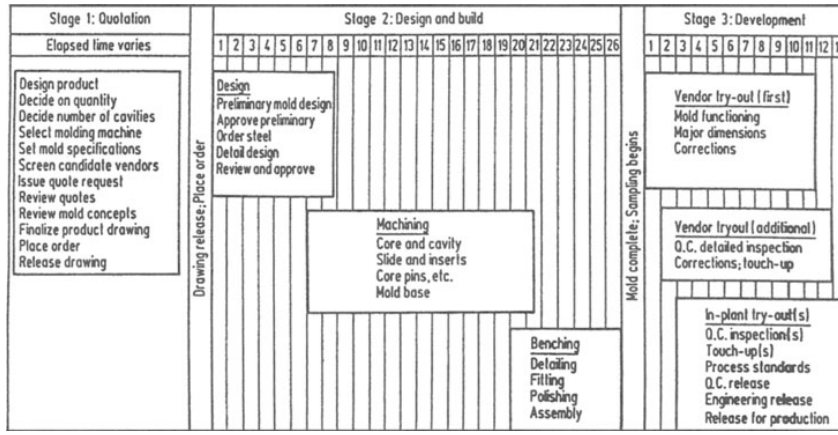
Mass production of standard elements for molds on large, highly efficient special machinery guarantees high quality and risk-free purchase because of fixed prices and reliable delivery dates.

Industry Guide

As in many major industries, various commercial and administrative practices have developed over the years that play an important role in the conduct of day-to-day business. These arrangements, generally expressed in the proposal, acknowledgment, and contract forms of individual companies, have been viewed as constituting "customs of the trade." An example is Table 4-29, which provides a guide to the events that usually occur in producing a mold.

DNA Genotek Inc. Exhibit 2029 Page 196
Spectrum Solutions LLC v. DNA Genotek Inc. IPR2023-01424

Table 4-29 Time guide to produce a mold; numbered columns represent weeks



The Moldmakers Division of SPI has a bulletin on moldmaking as part of its continuing effort to improve service to molders. It is intended to assist buyers seeking guidance in mold procurement. It points out the various difficulties that can result unless thorough understanding and communication are established between the mold buyer (molder) and moldmaker. Table 4-15 provides a guide for mold quotation.

Purchase Order

Once the decision has been reached to place an order with a mold builder, it is common practice to place the purchase order and its number by phone with a followup in writing. It is not good practice to use only telephone orders, which often lead to misunderstandings and unneeded delays. The order should include:

The mold specification sheet
Total price
Firm date of shipping (not ASAP)
Terms of payment
An acknowledgement copy

The acknowledgement copy of the purchase order should be received within seven days of the date when the order was placed. This will confirm to the buyer that the mold builder agrees to the specifications and conditions of the order or his or her exceptions to them.

To help ensure that the mold project stays on schedule, some mold builders have charts that show important milestones in mold construction. These milestones are the start and completion of:

Design
Models and hobs
Cavities
Cores and inserts
Mold base
Mold polish
Mold assembly

An example of a detailed mold progress report listing these milestones is shown in Table 4-30.

Table 4-31 provides a general mold progress report. The amount of detailed information required and frequency of the report are usually agreed on when the order is placed.

Mold Design

It is good practice to request a preliminary layout of the mold while it is being designed. To avoid delays, the layout should be reviewed and returned promptly with

DNA Genotek Inc. Exhibit 2029 Page 197
Spectrum Solutions LLC v. DNA Genotek Inc. IPR2023-01424

Table 4-30 Detailed mold progress report

COMPANY NAME

Address

+MOLD PROGRESS REPORT+

Part Name _____ P.O. Number _____ Job Number _____

Customer _____ Scheduled Delivery: Original __/__/__ Date of Report __/__/__

Attention of: _____ Current __/__/__ Report by: _____

Estimated Completion Week	DESIGN __/__ % complete	MODEL/HOBS __/__ % complete	CAVITIES __/__ % complete	CORES/INSERTS __/__ % complete	MOLD BASE __/__ % complete	POLISH __/__ % complete	ASSEMBLY __/__ % complete
1. __/__							
2. __/__							
3. __/__							
4. __/__							
5. __/__							
6. __/__							
7. __/__							
8. __/__							
9. __/__							
10. __/__							
11. __/__							
12. __/__							
13. __/__							
14. __/__							
15. __/__							
16. __/__							

Table 4-31 General mold progress report

TO: Date: __/__/__

JOB NUMBER _____ YOUR P.O. NUMBER _____

DESCRIPTION: _____

_____

_____

SCHEDULED DELIVERY: Original Date: __/__/__

Current Date: __/__/__

COMMENTS: As of this date, the mold is _____% complete.

_____

_____

_____

_____

_____

Signed: _____

Title: _____

_____

Table 4-32 Mold design checklist

1. ——	Was latest issue part drawing used?
2. ——	Will mold fit press for which intended? Are press ejectors specified?
3. ——	Are daylight and stroke of press sufficient for travel and ejection?
4. ——	Are reverse views correct?
5. ——	Are one guide pin and one return pin offset?
6. ——	Do guide pins enter before any part of mold?
7. ——	Can mold be assembled and disassembled easily?
8. ——	Has allowable draft been indicated?
9. ——	Are plastic material and shrinkage factor specified?
10. ——	Are mold plates heavy enough?
11. ——	Are mold parts to be hardened clearly specified?
12. ——	Are sufficient support pillars located and specified?
13. ——	Are waterlines, steam lines, thermocouple holes, or cartridge holes shown and specified?
14. ——	Does water in/out location clear press tie-bars and clamp locations?
15. ——	Is ejector travel sufficient?
16. ——	Are stop buttons under ejector bar specified?
17. ——	Are ejector pins sufficient? Specified?
18. ——	Is the steel type for mold parts specified?
19. ——	Have eyebolt holes been provided?
20. ——	If stripper type, does stripper plate ride on guide pins for full stroke?
21. ——	Do loose mold parts fit one way only? (Make foolproof.)
22. ——	Will molded part stay on ejector side of mold?
23. ——	Can molded part be ejected properly?
24. ——	Have trademarks and cavity numbers been specified?
25. ——	Has engraving been specified?
26. ——	Has mold identification been specified?
27. ——	Has plating or special finish been specified?
28. ——	Is there provision for clamping mold in press?
29. ——	Are runners, gates, and vents shown and specified?

any questions or comments. Questions you should consider with a preliminary layout are:

1. Will the mold fit the intended press, not only between the tie bars, but also between the minimum and maximum mold height?
2. Are press ejector holes shown?
3. Is sufficient ejection stroke provided?
4. Are there enough ejector pins, and are they placed properly?
5. Will the part stay on the ejection side of the mold?
6. Is there sufficient temperature control?
7. On side-draw molds, is there enough stroke to permit part ejection?
8. Is the cavity outline the reverse of the part?
9. Is there sufficient mold base steel supporting the cavities and cores on all sides?

As soon as the mold design is complete, two copies should be sent to the mold buyer: one for the buyer's file, the other to be returned to the mold builder with an approval signature. A design checklist is shown in Table 4-32.

Production of Molds

Most companies that produce injection molds are small- and medium-sized operations. Moldmaking shops range in size from a few individuals up to a maximum of about 200 employees. The exact number of shops and their size is not known, since many are not listed as moldmakers because they operate within another company or are classified under another part of the industry.

DNA Genotek Inc. Exhibit 2029 Page 199
Spectrum Solutions LLC v. DNA Genotek Inc. IPR2023-01424

The design departments of these companies are usually small; a scheduling department is often found only in the earliest stage of existence or not at all. A consequence is that, even for more complex molds, the design is often documented only in assembly drawings. Detailing and complete dimensioning of all individual components are often dispensed with for reasons of time and capacity. Inadequate planning at the design stage leads to considerably increased expenses during the production stage. Only the know-how of the personnel in the moldmaking shop and the absence of a distributed, industrial production process permit functionally relevant dimensions to be taken from assembly drawings and molds to be made without individual part drawings and work schedules. This leads to increased expenses for modifications during production and the risk that the drawings are not even corrected to incorporate these changes. This state of affairs results in further problems when producing replacement molds at the end of the service life of the original mold.

Mold Storage

During both short- and long-term storage (from hours to months or longer), steel molds must be protected from water and humidity. Unprotected steel can almost immediately begin to corrode, resulting in damaged molds that will require repolishing, regrinding, and/or repair at least of the surface. The result is cost in both labor and machine downtime. It is most cost-effective to protect the molds. There are excellent rust protectants on the market that operate for different time periods. However, most of these anticorrosion treatments must be completely removed before using the molds. Some may require special cleaners, including toxic solvents. Some operations dry off the mold and enclose it in an air-evacuated container.

Computer-Aided Mold and Product Design

Mold designers and builders can benefit from the use of CAD, CAM, and CAE techniques (Chap. 9). Computer programs permit analyzing the flow of plastics into cavities, designing mold cooling systems, determining mechanical stress in the molded product, etc. (175). These programs can simplify the design of molds with lower stress levels, less warpage, shorter cycle time, etc.

Additional aids such as lists of components used repeatedly are employed only rarely, although the range of components used and construction of the mold are well suited to such aids. The use of standard mold components represents an exception in this regard. Today, the extensive range of products offered by manufacturers of standard mold components is already employed whenever possible. In this way, the efforts of design and scheduling can be focused on product-specific shapes and dimensions. The use of CAD systems in conjunction with a standard mold component database can improve this situation even further.

CAD systems are found occasionally in the moldmaking industry. After an appropriate familiarization period, design work can be completed more quickly, but it is not in the preparation of drawings by means of a CAD system that the decisive advantage for moldmaking is most likely to be found. Only a complete CAD/CAM solution will permit the full economic benefits to be realized in the production of molds. Individual yet noteworthy examples already show today that in moldmaking, mold components can be produced in an industrial manner with a degree of automation ranging from minimal operator involvement to fully automatic production. In order to achieve this in general, however, it is necessary to depart from the currently encountered organizational and production structures.

Production Control Systems

The manufacture of a product, especially when carried out on an industrial scale, is subject to a range of requirements, the fulfillment of which is decisive to success of the market. They basically involve: (1) efficiency of the production process, (2) manufactured quality

DNA Genotek Inc. Exhibit 2029 Page 200
Spectrum Solutions LLC v. DNA Genotek Inc. IPR2023-01424

and performance, and (3) meeting delivery dates. In many plants, economic efficiency has always been the decisive factor in the startup of a product, but other factors have to be considered, such as ensuring that just-in-time (JIT) delivery occurs, the product is saleable, etc. A production planning and control (PPC) system has to cope with these requirements. General PPC systems are offered within the scope of commercial electronic data processing (EDP) programs for the operational sequences (Chap. 9).

A vital step in the design of a product is to determine if it will be capable of performing the task for which it is being designed, and what safety factor is available. This requires analysis and/or testing. The key area on which most analysis is focused is the mechanical load-bearing function for both tensile and compressive stresses. Valuable design equations are available in standard texts on designing with plastics and reinforced plastics, and the mechanics of materials can often be applied, depending on the product geometry. Such calculations can yield excellent predictions of short-term (dynamic) loading capabilities, as well as long-term (creep-related) approximations (Chap. 12). A key factor is to anticipate the extremes of temperature that can be encountered, especially high temperatures. Generous safety factors may be required in order to compensate for a variety of factors that can reduce the allowable load under extreme operating conditions (18).

Computer Monitoring of Information

Monitoring the available information on a fabricating machine is critical to improving its productivity. The value of such monitoring depends on the speed and ease with which data about a machine's performance can be integrated with the other information to provide a basis for control decisions. Transducers, servo controls, and other devices can provide the data needed for diagnostics and production monitoring, increasing the machine's productivity. This information can then be provided through the factory network (Chap. 9).

Productivity and People

People are needed to operate the plant efficiently. Machines, process controls, upstream and downstream equipment, design of parts, material handling, and all the technical and organizational elements in the plant (Fig. 1.1) can only operate efficiently if people set the plant into its correct pattern.

The recipe for productivity in any company includes a list of ingredients: research and development, new technologies, updated machinery, automated systems, and modern facilities, to name a few. But the one ingredient that ties the recipe together is people; none of the other factors has much impact without the right individuals. Without people who can do the research, who know the technologies, and who can use the machinery, you are not going to be productive no matter how large your capital expenditures are.

Management controls are only as good as the input they receive. To operate efficiently, one must understand how to obtain the maximum performance for each individual operation and—what is very important—to integrate the steps properly through planning. In that way, molded products that meet performance requirements at the lowest costs are produced.

Productivity and people interrelate with training. Throughout this book advice on training is provided particularly in Chap. 1 (Training Programs), Chap. 2 (Molding Operation Training Program), Chap. 9 (Software and Database Programs), and Chap. 12 (Training and People).

Value Analyses

Immediately after the molded product goes into production, the next step (a very important one) is to use the *value engineering analysis* approach: Produce parts that will meet the same performance requirements but are molded at a lower cost. There has to

DNA Genotek Inc. Exhibit 2029 Page 201
Spectrum Solutions LLC v. DNA Genotek Inc. IPR2023-01424

be room to reduce costs. If you do not take this approach, your competitors are sure to do so (Fig. 1-1).

Reevaluate all the parameters used in part design. Use less plastic, or use a lower-cost plastic with similar processing costs; or, very important, use a plastic with a higher cost that processes much faster, resulting in a total lower cost. Check hardware performance and all the other parameters described in this book.

The trouble with value analysis (VA) is that it sounds too good to be true. Thus, too many people give it little more than lip service. But it is good, and it is true. VA is like money in the bank and very often that *is* the problem. Many VA programs are set up to provide guaranteed savings rather than earn a maximum return on investment. Or to phrase it another way, VA is an organized study of function—but with some programs a little more organized than others.

Value analysis is the most effective, all-purpose technique in your professional tool kit. It is not exclusively a cost-cutting discipline. With VA, you literally can do it all: reduce costs, enhance quality, and boost productivity.

Value analysis sounds too easy. Like sports, singing, and writing, we all think VA is something we are naturally good at. Not so. For real results, VA (like the other three) demands hard, disciplined work. It must be a systematic, formal effort, endorsed and strongly supported by top management.

Here is a fast self-test, which is published by *Purchasing Magazine* annually to help you determine whether your department in fact has a working VA program. The questions are:

1. Is your top management committed to VA? Is there a written statement spelling out that commitment?

2. Does the person who heads up your program have any formal VA training?

3. Do your VA teams include people from a variety of departments and disciplines?

4. Have key members of your VA teams received any formal value analysis training?

5. How are VA projects or targets chosen?

6. Are progress reports made on team meetings?

7. Is the emphasis consistently focused on function?

8. Is there a VA manual? If not, is there a VA section in the purchasing manual?

9. What VA targets or goals have been set over the past several years? What were actual results?

10. Do you look for VA-oriented suppliers?

11. Is supplier VA help encouraged? Are suppliers included on VA teams? Are they rewarded for their contributions?

12. Is your program a continuing effort, or is it a crash cost-cutting response to bad times?

Zero Defects

All targets in any area of industries worldwide, that includes the plastics industry, are for zero defects. From the concept to the production of a product, different actions can be taken to ensure meeting the target or coming as close to it as possible. Zero defects can equate to the ultimate performance for any molder or supplier in terms of quality or producing a product to meet design performance requirements at the lowest cost.

This is unlikely to be achieved by the usual quality control procedures, since QC analysis is usually only made after production and inherently is based on the acceptance of a certain level of failure. Therefore, if a quality standard is to be really effective, it must start earlier than the production shop. Everyone from top management to those in production must think quality and realize that any acceptance of a second-best attitude is not permissible (Chaps. 12 and 13).

Terminology

Adapter plate The plate holding the mold to the molding machine press or platen.

Back draft A term sometimes used to describe a detail of a molding that is smaller than the normal mouth opening of the mold. The opposite is a mold undercut.

Backing plate A plate used to support cavity blocks, guide pins, bushings, and similar mold parts.

Bluing A mild blemish in the form of a blue oxide film which occurs on the polished surface of a mold as a result of the use of abnormally high mold temperatures.

Bluing off Checking the accuracy of mold cutoff surfaces by putting a thin coating of Prussian blue on one half and checking the blue transfer to the other half. Other techniques used include carbon paper, shims, etc.

Bolting pattern, mold to platen The SPI standards specify the location and size of tapped holes in the stationary and moving platens for the attachment of molds.

Bottom plate Part of the mold containing the heel radius and pushups (ejection mechanism). It is used to join the lower section of the mold to the platen.

Breathing Also called mold bumping, dwell pause, dwell, gassing, and degassing. It is a pause in the application of mold pressure to allow the escape of gases formed by certain plastics during the heating process; also to remove any entrapped air. This on–off–on pressure action occurs just prior to having the mold completely closed. Materials that require breathing include many TS plastics, and TS elastomers and rubbers during vulcanization.

Cam bar The stationary angled bar or rod used to mechanically operate the slides on a mold for side-action core pulls.

Cavity The space between matched molds that encloses the molded part. It is the depression in the mold that forms the outer surface of the molded part. There can be single or multiple cavities in one mold.

Cavity chase An enclosure of any shape, used (1) to shrink-fit parts of a mold cavity in place, (2) to prevent spreading or distortion in hobbing, or (3) enclose an assembly of two or more parts of a split cavity block.

Cavity, compression A male cavity designed as a plug that fits into the female cavity so that the mold action during closing provides hydraulic pressure loading. The tight-fitting male plug acts as a hydraulic ram.

Cavity, debossed A cavity with depressed (indented) lettering or designs producing bossed impressions on the molded part.

Cavity deposit Plastic buildup on a cavity's surface due to plate-out of the plastic; usually attributed to the use of certain additives.

Cavity duplicate plate A removable plate that retains cavities; used where two-plate operation is necessary for loading inserts.

Cavity, etched A cavity whose surface has been treated with an acid, leaving relief to form the desired design texture on the molded part.

Cavity ejector Any of various mechanical means used to eject the molded part from the cavity.

Cavity, female The indented half of a mold designed to receive the male half.

Cavity grit blasting Steel grit or sand is blown onto the cavity wall to produce a rough surface. This surface treatment may be required to permit air to leave the mold during molding and/or provide a desired surface finish on the part.

Cavity hobbing Forming single or multiple mold cavities by forcing a hob into a relatively soft steel blank. A master model in hardened steel is used to sink the shape of the cavity into a heated mild steel such as beryllium copper. The hob is larger than the

finished plastic molded part, because after hobbing, the metal shrinks during cooling.

Cavity honing Using a fine-grained whetstone or equivalent to obtain precise accuracy of surface finish.

Cavity land A region in a gate configurations that controls melt flow.

Cavity, male Also called plunger. The extended half of a mold designed to match the female half.

Cavity pressure The cavity pressure can be recorded via a transducer located, for example, in the cavity near the gate. It can plot a profile that records different information such as filling, packing, and holding pressures.

Cavity register Angle faces on the mold that match when the mold halves are closed, to ensure their correct alignment.

Cavity retainer plate Any of the plates that hold the inserted cavities in a mold. These plates are at the mold parting line and usually contain the guide pins and bushings that line up the two halves of the mold.

Cavity side part In the United States, the stationary part of an IMM. In the U.K., the side of the injection mold that is adjacent to the nozzle.

Cavity, split A cavity made in sections.

Cavity, split-ring A mold in which a split cavity block is assembled in a chase to permit the forming of undercuts in a molded part. It is ejected from the mold along with the molded part(s) and then separated.

Cavity unit Cavity insert(s) designed for quick interchangeability with other cavity insert(s).

Cavity venting Shallow channel(s) or minute hole(s) in the cavity and/or in the mold parting line to allow air and other gases that may form during processing to escape.

Chase See *Cavity chase.*

Chase floating A mold member, free to move, that fits over a cavity or a lower plug, and into which an upper plug telescopes.

Chrome plating Also called chromium plating or Cr plating. An electrolytic process that deposits a hard, semigray film of chromium metal onto properly prepared surfaces of materials (mold cavity, plastic parts, etc.). Chrome-plated surfaces are frequently used where resistance to corrosion or abrasion is needed, as in molds and other tools. When it is used as a plastic coating, decorative effects are obtained.

Chunk An open-face mold.

Cold molding A properly prepared compound is shaped at room temperature in a mold and subsequently cured by heating (baking) in an oven.

Cold slug The first thermoplastic melt to enter an IMM cold-runner mold; so called because in passing through the sprue orifice it is cooled below the effective molding temperature. Usually a well in the runner system is used to unload the cold slug.

Cold-slug well The space or cutout in the runner system (e.g., opposite the sprue travel of the melt in the mold) to trap the cold slug so that it enters the cavity.

Compression flash ring The ring of excess melt that escapes from the cavity into the clearance between the force male plug and the vertical or horizontal wall of the female cavity in a positive or semipositive mold.

Compression force Also called the punch, plunger, or ram. The male half of the mold that enters the cavity and exerts pressure on the plastic, causing it to flow.

Compression mold A typical two-part mold that has a female cavity with a matching male plug cavity that fits into it. In the

closed position, the molding material is compressed in the space between the mating cavities. With the usual TS plastic used, flash occurs. This type of mold is used for injection–compression molding (coining).

Compression mold, positive A compression mold designed to apply a constant pressure to the part being molded, the thickness of the part being determined by the amount of charge. The mold is designed to trap all the molding material when it closes.

Compression mold, semipositive A combination of positive and (vertical or horizontal) flash compression mold. It operates as a flash mold until within a short distance of the final closure, when the force plug telescopes within the chase to exert a positive pressure on the charge during the final closing of the mold.

Compression plastic material well Space provided in the male cavity to handle the bulk of the material being loaded in the female cavity.

Compression shear edge A telescoping shear edge located around the periphery of the mold cavity for RP materials (SMC, etc.). Its functions are to seal off the mold when closed, release or vent air and gases from the mold cavity, and/or permit the cavity half of the mold to slide over the core half so the required pressure can be applied to the material. To control part dimensions, it is important to control the size (weight and volume) of the material charge.

Core A channel in a mold for circulation of a heat-transfer medium. It is part of a complex mold that molds undercut parts. Cores are usually withdrawn to one side before the main sections of the mold open. They have passages for heat transfer to the melt in the cavity.

Cored mold A mold incorporating ducts which permit the passage of heating and cooling fluids.

Core pin A pin used to mold a hole in the molded part.

Core-pulling sequence The SPI recommended core-pulling sequences are as follows: (1) sequence A (clamping required only with mechanical ejector) with reset ejector, core-in, clamp close, inject, clamp open (continue), and eject; (2) sequence B with clamp close, cores-in, inject, cores-out, clamp open, and eject; (3) sequence C (can only be used in hydraulic ejection) with lamp close, inject, clamp open, cores-out, eject, and cores-in; and sequence D (requires interlock to ensure cores are in proper position prior to injection or ejection) with clamp close during cores-in, inject, clamp open during cores-out, and eject.

Core, side Also called side draw pin or cam pin action. Projections that are used to core a hole (or other shape) in a direction other than the line of closing of a mold. It is withdrawn before the part is ejected and/or prior to the mold opening.

Deflashing Removal of flash from a plastic product, usually a molding. Several different methods are employed, including low temperature (cryogenic).

Deflashing, cryogenic Deflashing parts, particularly when small and numerous, can be done efficiently using cryogenic tumblers and shot blast. Liquid nitrogen [at −320°F (−196°C)] or dry ice is used. After chilling, the parts can be blasted, usually with a plastic, while tumbling in a basket sealed in an enclosed chamber. The tumbler air-moving system can be sprayed at about 225 psi (1.6 MPa). Advantage of this procedure includes accuracy, repeatable deflashing, and reduced finishing costs.

Deflashing, pressure blasting This method utilizes a stream of small pellets, usually crushed fruit pits or other particles that are not as hard as the plastic being deflashed, thrown at high speed at the molded parts, which are tumbling over a continuous

DNA Genotek Inc. Exhibit 2029 Page 205
Spectrum Solutions LLC v. DNA Genotek Inc. IPR2023-01424

moving belt or in a perforated container or basket.

Deflashing, wheelabrator Deflashing molded and other parts by bombarding with small particles at high air velocity.

Degating Separating the molded part from the runner system, automatically or manually, in or out of the mold.

Dehumidification If you use chilled water as a heat-transfer medium to reduce the cycle time, then during high-humidity periods you may have to deal with condensation forming on the surface of the mold, which usually causes imperfections on the molded part. Remedies include the use of an enclosure filled with dehumidified air, and the application of a high-velocity dry airstream.

Die-slide molding A patented process from Japan Steel Works for injection molding two halves of a hollow part in cavities on opposite sides of a single mold. After the mold opens, a slide plate on the stationary platen aligns the two parts and a second injection joins the parts.

Dished A term used to describe a molded surface having a shallow depression.

Double-shot molding A method for incorporating two colors or two different plastics in a part, using an IMM with two plasticators. The part molded first becomes an insert for the second shot. Alternative processes include injection blow molding and compression molding.

Dowel Also called mold pin or retaining pin. A metal pin located in one half of a mold that enters a corresponding hole in the other half so that, upon closure of the mold, the two halves become correctly aligned.

Dowel bushing A hardened steel bushing lining a dowel hole.

Dwell A pause in pressurization just prior to the mold completely closing. It is the time between when the injection screw ram is fully forward, holding pressure on the plastic in the cavity, and the time the ram retracts.

Ejection mark A surface mark on the part caused by the ejector pin when it pushes the part out of the mold cavity. It may need to be located where it is acceptable.

Feed bushing A hardened steel bushing in an injection mold that forms a seal between the mold and the injection unit.

Film insert molding A method that starts with a cut film, which is decorated and/or labeled, thermoformed to shape, and then inserted in the mold.

Flash mold A mold whose land surface permits the escape of excess molding material and has no trimming action. Such a mold relies upon back pressure to seal it and put the part under pressure.

Flash groove A groove ground in the parting-line land to allow the escape of excess plastic during the molding process.

Flash line A raised line evident on the surface of a molding and formed at a junction of mold faces, as at the parting line, after the removal of the excess flash. It is usually removed by high-speed buffing or grinding.

Flash trap A molded-in lip or blind recess on a part that is used for trapping excess melt (flash).

Frozen layer Plastic melt begins to freeze (solidify) as it starts filling an injection mold cavity. The frozen layer can easily vary in thickness as the mold fills, producing different frictional shear forces. As a result, flow (filling) and solidification (TP cooling) should be evaluated together.

Grid The array of channel shaped supporting members within a mold.

DNA Genotek Inc. Exhibit 2029 Page 206
Spectrum Solutions LLC v. DNA Genotek Inc. IPR2023-01424

Hold-down groove A small groove cut into the side wall of the mold cavity surface to assist in holding the molded part in the cavity while the mold opens.

Inching Reduction in the rate of mold closing travel just before the mating mold surfaces touch each other.

Knife-edge A projection from the mold surface that has a narrow included angle. Knife-edges are undesirable because they are susceptible to breakage and/or wear under molding pressures.

Land An area where the faces of a closed injection mold come into contact with one another.

Latch A mechanical device to hold together two members of a mold.

Latch plate A plate that retains a removable core to hold an insert carrying pins on the upper part of the mold.

Leader pins and bushings Also called guide pins. Pins (usually four) to maintain the proper alignment of the male plug and female cavity as the mold closes. One of the pins is not symmetrically placed, so that the mold halves can only be aligned one way, eliminating misalignment. Hardened steel pins fit closely into hardened steel bushings.

Lift A complete set of moldings produced in a single operation of an IMM. The output rate may be expressed as the number of lifts per hour.

Loading well A volume in the top of a cavity, usually for molding bulky compounds. Its size is dependent on the material's bulk factor.

Locating ring Also called register ring. A ring that serves to align the nozzle of an injection cylinder with the entrance of the mold's sprue bushing.

Lubricant A substance applied on or injected into molds to eliminate or reduce friction and/or prevent adhesion of its component parts.

Manifold A runner system in a mold, which can have its own insulated heating and/or cooling section, to control the melt and make it ready for injection into the cavity.

Meld line Similar to a weld line, except the melt flow fronts move in parallel rather than meet head on (perpendicular). See *Weld line.*

Melt extractor A device, such as a spreader (torpedo), which is placed in a plasticizing system for the purpose of separating fully plasticated melt from partially molten plastic.

Mold base standards The SPI continually updates its publication on designing plastic molded parts entitled Standards and Practices of Plastics Molders. It is useful to designers, purchasing agents, custom molders, processors, etc. It includes engineering and technical guidelines commonly used by molders for injection, compression, and transfer molding processes; lists tolerance specifications for plastic materials in metric and English units; and provides a glossary of terms. It reviews important commercial and administrative practices for purchasers to consider when specifying and purchasing molded parts. These customs of the trade include mold type, safety considerations, maintenance requirements, contract obligations, charges and costs, inspection limitations, storage, disposal, proper packing and shipping, and claims for defects.

Mold, controlled-density A mold with a variable-volume cavity incorporating either a movable wall or a plug-type section in the wall. After filling the cavity with melt, the cavity is compressed by the movable wall or plug. With foam molding, controlled solid skins with core are achieved. This is a modification of the injection–compression technique.

Mold cooler partitioned A large-diameter hole drilled into the mold (usually the core) and partitioned by a metal plate extending to near the bottom end of the channel. Water is introduced near the top of one side of the partition and removed on the other side. This device is like a bubbler.

Mold cooling Cooling of the mold (for thermoplastics) is an essential mold feature and requires special attention in mold design. The cooling system should ensure rapid and uniform cooling of the molding. In the design of mold components and layout of guides and ejectors, allowance should be made for the proper size and positioning of the cooling system. Rapid cooling improves process economics, whereas uniform cooling improves product quality by preventing differential shrinkage, internal stresses, and mold release problems. In addition, uniform cooling ensures a shorter molding cycle.

Mold-cooling and -heating channels Passageways (usually drilled holes) located within the body of the mold through which a turbulent fluid (cooling medium) can be circulated to control the temperature on the mold cavity surface. They may also be used for heating a mold by circulating heated fluids (oil, steam, etc.), as in the molding of thermoset and some thermoplastic materials. (Heating can also be accomplished using electric heaters in the mold body.) There are applications where the press platens are heated and in turn heat the mold or (more often) directly heat flat plastic-laminated material such as thermoset decorative panels, printed circuitboard panels, reinforced plastic building panels, etc.

Mold-cooling channel bubbler A device inserted into a mold cavity, such as a rib or core, that allows water to flow deep inside the hole into which it is inserted and to discharge through the open end of the hole. Uniform cooling of the mold and isolated mold sections can be achieved in this matter.

Mold-cooling flooding Molds, particularly for blow molding, can use a box-type enclosure next to the cavity wall rather than pipe passageways. This flood-type turbulent mold-cooling system is less expensive and provides adequate cooling.

Mold cooling, spiral method A method of cooling injection molds or similar molds in which the cooling medium flows through a spiral cavity in the body of the mold. The cooling medium is introduced at the center of the spiral, near the sprue section, because more heat is localized in this section.

Mold cooling time In addition to the mold, plastic material, and machine costs, the final cost to mold a part depends on the molding cycle. A large part of this cycle, up to 80%, is due to the time required to cool the molding. This time depends on the heat of the molding. The minimum cycle time, therefore, is governed by the time taken to cool. The injected plastic is cooled rapidly by its contact with the cavity wall, but since plastics are poor heat conductors, the solidified outer layer retards heat transfer from the center of the molding. Most of the cooling time is thus required to cool this center. Often, the molding may be released from the mold as soon as its outer layer is sufficiently rigid. This temperature is called the mold-release temperature. The inside of the molding will often still be considerably hotter. The minimum cooling time required to reach mold-release temperature is governed by the wall thickness of the molding, the difference between the polymer and mold temperatures, and that between the mold-release temperature of the article and mold temperature.

Mold-cooling vacuum Rather than pushing liquid coolant through a mold, pulling the coolant can provide advantages such as eliminating water leaks and reaching complicated cavity surface areas to provide cooling action by using waterline venting.

Mold cut-off Also called shutoff or flash land. The part of the mold land that isolates the molding.

Mold, deep draw A mold having a core that is appreciably longer than the wall thickness.

Mold deformation After the molding pressure stroke and during any afterfill, pressure is built up in the mold cavity. During injection molding this pressure is generally one-third to one-half of the pressure in the IMM plasticator. Such pressure can cause elastic deformation such as bending of the cavity retainer plates and cores, ejector and guide pins, etc. To reduce this action, sturdy construction of the mold is required. This conflicts with the desire to minimize the amount of mold material for efficient cooling.

Mold degating Separating the molded part from the runner system, automatically or manually, in or out of the mold.

Mold design Computers are used in many designs. Views, cross sections, projections, changes in size and color, and mechanical and thermal analyses are then easily made.

Mold, duplicating A mold made by casting over (duplicating) another product by mechanical reproduction using cutting tools that are guided by a master, proportional in size to the desired finished products.

Molded edge An edge that is not physically altered after molding (with fiber reinforcements) for use in final form, particularly one that does not have fiber ends along its length.

Molded net A means of describing a molded part which requires no additional processing to meet dimensional requirements.

Molded Parts Buyers Guide This guide has been prepared by the Molders Division of the SPI. It contains important points that purchasers have traditionally considered in specifying and purchasing plastic parts. As in every major fabricating industry, various commercial and administrative practices have developed over the years that play an important role in the conduct of day-to-day business. These arrangements, generally expressed in the proposal, acknowledgement, and contract forms of the individual molding companies, have been viewed as constituting "customs of the trade." This informative manual is designed to identify and explain these customs.

Mold efficiency In a multimold blowing system, the percentage of the total turnaround time of the mold actually required for forming, cooling, and ejection of the blown part.

Mold ejection A device or system fitted to (usually) the moving platen of a machine for operating the molding ejector(s) to remove molded parts. It may be operated mechanically (e.g., with springs), hydraulically, pneumatically, or electrically. It operates in sequence with the clamping close preposition, a provision in the clamping unit that actuates the ejection action. Typical means are knockout pins, stripper plates or rings, unscrewing, cams, removable inserts, or bushings. The choice of ejector system is largely governed by the article shape and by the rigidity or flexibility of the plastic used. The mold should preferably be fitted with ejectors at those spots around which the molding is expected to shrink (e.g., around cores). At high mold temperatures, allowance must be made for thermal expansion of the mold platens. These platens will expand more than the plates of the ejector mechanism. It is therefore recommended that the ejectors be provided with a cylindrical head and mounted with some clearance to allow the correction of possible variations in center distances during machine operation. The ejection of articles with large cylindrical or flat surfaces may sometimes be hampered by the creation of a vacuum between the article and the cavity wall. In such cases, release may be improved and the vacuum broken by an air ejection system.

Mold ejection mark See Ejection mark.

DNA Genotek Inc. Exhibit 2029 Page 209
Spectrum Solutions LLC v. DNA Genotek Inc. IPR2023-01424

Mold ejector pin A rod or sleeve that pushes a molding off a core or out of a cavity mold. It is attached to an ejector bar or plate. It is also called a knockout pin.

Mold ejector plate See *Mold ejector pin.*

Mold ejector ram A small hydraulic, mechanical, or electrical ram fitted to a molding press for the purpose of operating ejector pin(s).

Mold ejector retainer plate A retainer in which one or more ejector pins are assembled.

Mold ejector return pin One of the projections that push the ejector assembly back as the mold closes. It is also called a surface pin, return pin, safety pin, or position pushback.

Mold ejector rod or bar A bar that actuates the ejector assembly when the mold is opened.

Mold ejector sleeve A bushing-type ejector.

Mold ejector spider A system where part of an ejector mechanism operates the ejector pin(s).

Mold, elastomeric Elastic or stretchable mold made of rubber (elastomer), rather than the usual steel, so that complex shaped parts can be removed without mold side actions, etc. Usually used for casting plastics. They can be stretched to remove cured parts having undercuts, etc.

Mold, family Sometimes called a combination mold. A multicavity mold where each of the cavities forms one component part of the assembled finished product. The term is also often applied to molds where parts from different customers are grouped together in one mold for economy of production.

Mold feed bushing The hardened steel bushing in an injection mold that forms a seal between the mold and injection nozzle.

Mold flash A thin surplus web of plastic material, usually occurring with thermoset plastics, attached to a molding along the parting lines, fins at holes or openings, etc. With most moldings, it must be removed before the parts are acceptable.

Mold flash groove By this term, we mean a groove ground in the parting-line land to allow the escape of excess material during the molding operation, particularly compression molding.

Mold flash line See *Flash line.*

Mold flash ridge The part of a flash compression mold through which the excess material escapes until the mold is closed.

Mold flash ring, vertical or horizontal The clearance between the force plug and the vertical or horizontal wall of the compression molding cavity in a positive or semipositive mold; also, the ring of excess material that escapes from the cavity into this clearance space.

Mold force That portion of a mold which forms the interior of the part; sometimes called a core or plunger. In compression molding, the downward-acting mold half, usually the male half.

Mold force plate The plate that carries the plunger (force plug) of a compression mold and the guide pins or brushings. Since it is usually drilled for hot water or steam lines, it is also called the hot or steam plate.

Mold force plug The male half of a compression mold that enters the cavity, exerting pressure on the plastic and causing it to flow. It is also called a core, plunger, or ram.

Mold, French A two-piece mold for irregular shapes—tall, top-heavy, leaning to one side, or with extremely fine detail.

Mold gate The orifice through which the melt enters the mold cavity. It can have a variety of configurations, depending on product design.

Mold grid An array of channel-shaped mold-supporting members.

Mold half One of the two basic parts to a mold. Each part is called a mold half, but that usually does not mean that the mold is divided dimensionally into two equal halves.

Mold, hand Also called portable or loose mold. A small mold that is removed by hand from the press for the purpose of stripping molded parts and/or reloading (plastic and/or inserts).

Mold heated-manifold A mold in which the portion (the manifold) that contains the runner system has its own heating elements, which keep the molding material in a melted state for injection into the cavities, from which the manifold in insulated.

Mold heat-transfer device A device that transfers localized heat to a heat sink in order to improve mold cooling, or heat from a heat source to a localized area such as hot sprue bushings.

Mold height The overall thickness of the mold between the platens of the molding machine. It is the height when the mold sits on a table. Thus, in a conventional IMM in operation, is the horizontal dimension between the platens.

Mold hobbing A technique in which a master model in hardened steel is used to sink the shape of a mold cavity into a softer material such as heated mild steel on beryllium copper. The hob is larger than the finished plastic part. After hobbing, the metal shrinks as it cools.

Mold hold-down groove See *Hold-down groove.*

Mold, hollow A mold that permits melted plastic to be applied to its inside surface to form hollow-shaped parts.

Mold inching See *Inching.*

Molding The forming or shaping of a plastic or reinforced plastic into a solid mass of prescribed shape and size by the application of pressure (zero on up) and in most processes heat for a given time.

Molding compounds Plastic material in varying forms (pellet, granulation, or gunk) consisting of plastics, filler, pigment, reinforcement, plasticizer, and/or other ingredients ready for molding. It is also called dry blend, molding powder, bulk molding compound, and sheet molding compound.

Molding cycle (1) The period of time required for the complete sequence of operations on a molding press to produce one set of moldings. (2) The operations necessary to produce a set of moldings, without reference to the time. The sequence of operation (manual, semiautomatic, or automatic) is (a) close and clamp the mold, (b) inject the mold, (c) hold mold closed under pressure while plastic cools or cures, (d) open the mold, and (e) eject the part.

Molding index The result of a test used with thermoset molding powder in which a standard flash-type cup mold under prescribed conditions is used. The molding index is the total minimum force required to close the mold (ASTM D 731).

Mold, interchangeable The SPI and other organizations have published standards in regard to mounting molds in the IMMs. The obvious advantage of these standards is that they easily allow molds to be designed to run on more than one brand of machines.

Molding pressure The pressure applied either directly or indirectly to the ram action of an injection machine, compression press, transfer press, etc. to force the melt to completely fill the mold cavity.

DNA Genotek Inc. Exhibit 2029 Page 211
Spectrum Solutions LLC v. DNA Genotek Inc. IPR2023-01424

Molding pressure, contact A method of molding or laminating in which the pressure, usually less than 70 kPa (10 psi), is only slightly more than necessary to hold the materials together during molding. It is also called contact molding especially in connection with reinforced plastics.

Molding pressure, high In a molding or laminating process, a pressure greater than 1,400 kPa (200 psi), but commonly 7 to 13.8 MPa (1 to 2 ksi). Most often used in connection with reinforced plastics.

Molding pressure, low The range of pressures from 2,760 kPa (400 psi) down to and including pressure obtained by the mere contact of the plies or material. Most often used in connection with reinforced plastics.

Molding pressure required The unit pressure applied to the molding material in the mold. The area is calculated from the projected area taken at right angles under pressure during complete closing of the mold, including areas of runners that solidify. The unit pressure is calculated by dividing the total applied force by this projected area and is expressed in psi or Pa.

To determine the pressure required on a mold that has a specific projected area based on the plastic to be used, the pressure required on the melt is determined (either from past experience or from the material supplier). This pressure (in psi or Pa) is multiplied by the total area. The result is total clamping force required. This force provides a guide to the clamp tonnage needed in the IMM; to be safe it is best to have 10% more available.

Molding, short An incomplete molding due to a short shot (insufficient plastic to fill the mold).

Molding, structural-web This low-pressure foam molding method bridges the gap between structural foam molding and injection molding. The surface of the part does not have the characteristic swirl pattern of the former method. Structural-web molding can produce very large, lightweight parts with smooth surfaces like conventional injection-molded parts.

Molding, two-shell A technique to produce hollow parts by molding (injection, compression, blow molding, rotational molding, etc.) two halves with mating flanges or the equivalent, and then assembling them by various techniques.

Mold insert That part of a mold cavity or force which forms undercut or raised portions of a molded product.

Mold knockout bar A bar that holds and actuates ejector pin(s) in a mold.

Mold land (1) In an extrusion die (sometimes called mold or tool), the surface parallel to the flow of material. (2) In a two-piece mold, a platform built up to the split line. (3) The portion of a mold that provides the separation (cutoff) of the flash from the molded part. (4) The bearing surface of a mold by which excess material escapes. (5) In a semipositive or flash mold, the horizontal bearing surface. (6) The nozzle region of a nozzle used in injection molding. (7) One of the parallel parts of a gate. (8) The bearing surface along the top of the flights of a screw in a plasticator.

Mold land area The whole area of contact, perpendicular to the direction of application of pressure, of the seating faces of a mold (those faces that come into contact when the mold is closed).

Mold land force A force with a shoulder that sits on a land in a landed positive mold. It is also called a landed plunger.

Mold latch See *Latch.*

Mold latch plate A plate used for retaining a removable mold core of large diameter, or for holding insert-carrying pins on the upper part of a mold. Release of the pins or core is effected by moving the latch plate.

DNA Genotek Inc. Exhibit 2029 Page 212
Spectrum Solutions LLC v. DNA Genotek Inc. IPR2023-01424

Mold leader pin and bushing The mating mold components used to align and guide the two halves of the mold as it opens and closes in the machine. Hardened steel leader pins are also called guide pins. The pins fit closely into hardened steel bushings.

Mold loading well The top area of a compression mold cavity, the size of which is dictated by the bulk factor of the molding compound. High-bulk-factor materials require deeper wells than low types.

Mold locating ring This is a ring that serves to align the nozzle of an injection cylinder with the entrance of the sprue bushing, and the mold with the machine platen. It is also called a register ring.

Mold locking force Refers to the force exerted in the locking mechanism of the machine that keeps the mold closed during injection.

Mold locking mechanism A hydraulic cylinder or toggle mechanism to close the mold and keep it in the closed position during injection.

Mold lubricant See *Lubricant.*

Mold manifold The configuration of piping in a block of metal that takes a single-channel flow of melt from a machine (extruder, injection, etc.) and divides it into various flow channels to feed more than one outlet.

Mold manifold, nozzle A series of injection nozzles mounted on a common manifold, each nozzle positioned so as to feed a single cavity in the mold. Such manifolds are used to eliminate runners in molds such as cup-shaped articles, when it is desired to gate the cavities at the centers of the bottoms.

Mold manifold shutoff valve A valve used to shut off plastic flow, usually mounted in the manifold.

Mold, mounting dimensions The SPI Injection Molding Division provides a guideline bulletin that recommends interchangeable mold mounting dimensions for various-size injection molding machines. It includes platen bolting patterns, tap hold threads, knockout pin locations, and sizes of nozzles and locating rings.

Mold number or mark The number assigned to each mold or set of molds for identification purposes. It is usually placed in a unobtrusive area such as that part of a container mold that forms the base of the container.

Mold orifice groove A small groove used in molds to allow material to flow freely to prevent weld lines and low density, and to dispose of excess material.

Mold parallel to the draw The axis of the cored position (hole) or insert is parallel to the up and down movement of the mold as it opens and closes.

Mold parting line A line established on a three-dimensional model from which a mold is to be prepared, to indicate where the mold is to be split into two halves (sections) or several components.

Mold pillar support The general construction of a mold base usually incorporates the U-shaped ejection housing. If the span between the arms of the U is long enough, the forces of molding can cause a sizable deflection in the plates that are supported by the ejector housing. Such a deflection will cause flashing of parts. To overcome this problem, the span between supports is reduced by placing pillar supports at certain spacings so that deflection is eliminated or negligible.

Mold pin (1) Mold dowel pin. (2) Mold ejector pin. (3) Mold leader pin. (4) Mold return pin. (5) Mold side draw pin. (6) Mold sprue draw pin.

Mold, porous A mold made up of bonded or fused aggregate (powdered metal, coarse

pellets, etc.) such that the resulting mass contains numerous open interstices of regular or irregular size, allowing either air or liquids to pass through the mass of the mold. Such molds are used particularly in thermoforming.

Mold, positive (1) A mold designed to trap all the molding material when it closes. (2) A projecting mold over which the part is thermoformed. This type is often referred to as a male mold. (3) A compression mold designed with vertical shutoff.

Mold pot To embed a component or assembly in liquid plastic, using a shell, can, or case that remains an integral part of the product after the plastic is cured. (2) A chamber to hold and heat molding material for a transfer mold.

Mold pot plunger A plunger used to force softened molding material into the closed cavity of a transfer mold.

Mold, preengineered Standardized mold components, such as ejector pins, guide pins, bolts, etc., and complete standardized mold assemblies have been commercially available since 1943. Advantages include exceptional quality control on materials used, low cost, quick delivery, interchangeability, and promotion of standardization.

Mold pressure pad One of the reinforcements of hardened steel distributed around the dead area in the faces of a mold to help the mold land absorb the final pressure of closing without collapsing.

Mold production and handling Different standards and practices of plastic molders are reviewed in the SPI Molders Division Guide bulletin. It includes the following: (1) mold maintenance, repair, and/or replacement; (2) molds on consignment; (3) mold drawing; (4) mold usage; (5) mold storage; (6) mold removal; and (7) amortization and insurance.

Mold, reentrant A mold containing an undercut that tends to resist withdrawal of the molded part.

Mold restrictor ring A ring-shaped part protruding from the torpedo surface that provides an increase of pressure in the mold to improve the welding of two melt streams.

Mold retainer pin (1) A pin on which an insert is placed in the mold and located prior to molding. (2) One of the pins that return the ejector mechanism to the molding position.

Mold retainer plate The plate on which demountable pieces, such as mold cavities, ejector pins, guide pins, and bushings, are mounted during molding; it is usually drilled for water or steam.

Mold retainer plate nest A retainer plate with a depressed area for cavity blocks used in injection molding.

Mold rod guide A rod that guides the platens but takes no clamp force.

Mold, rotary Also called rotary press. A type of injection molding, blow molding, compression molding, etc. utilizing multiple mold cavities mounted on a rotating platen or table. This process is not to be confused with rotational molding.

Mold, rotational Molds are manufactured from electroformed nickel, vapor-deposited nickel, and cast aluminum. The thickness of the molded parts is controlled by heat sinks fabricated into the tool. The tool must be temperature-controlled so it will cure the plastic within it.

Mold runner A groove or channel of any size, shape, and depth, through which the melt flows to the cavity(ies); this is the channel that connects the sprue to the gate. The term runner system is sometimes applied to all the material in the form of sprues, runners, and gates between the machine nozzle and cavity(ies). There are different types of

runners. The most popular are cold runners and hot runners, used with thermoplastic and thermoset plastics; others include insulated and stacked. With TP, in a cold runner the melt solidifies when the part solidifies; in the hot runner, it remains liquid. TS plastic experiences the reverse: With a hot runner, the melt in the runner solidifies, and with a cold runner, it remains a melt.

Mold runner, balanced In a multicavity mold, the runners are said to be balanced when the linear distances of melt flow from the sprue through the runner network to the gates of all the cavities are of equal length.

Mold runner, cold, for thermoplastic The sprue, runner(s), and gate(s) of the TP melt, like the melt in the cavity(ies), all solidify by the cooling action of the mold. This mold design produces solidified sprue and runner(s) that are usually granulated and recycled.

Mold runner, cold, for thermoset Such a runner provides for injection directly into the cavity from the gate. The runner manifold section is cooled to maintain plastic in a melt stage. The cavity and core plates are heated (to solidify plastic) to normal molding temperature and insulated from the cooler manifold section. This mold design eliminates TS scrap loss from sprue and runner(s), as in a hot-runner system for thermoplastics.

Mold runner, hot, for thermoplastic The sprue and runner(s) are insulated from the chilled cavities and remain hot, so that the runner never cools in normal cycle operation. Runners are not ejected with the molded part(s). Thus, the next shot is from the gate rather than the machine nozzle. An insulated runner is a type of hot runner.

Mold runner, hot, for thermoset The sprue, runner(s), and gate(s) of the TS melt, like the melt in the cavity(ies), all solidify by the heating action of the mold. This mold design produces solidified sprue and runner(s) that can be granulated and recycled at least as plastic filler.

Mold runner, insulated An oversized runner passage formed like a conventional cold runner for thermoplastic. Runner insulation is provided by a layer of chilled plastic that forms on the runner wall. The passages in the mold plate are of sufficient size that, under conditions of operation, the insulating effect of the plastic combined with the heat applied with each shot maintains an open path.

Mold runner, internal A hidden flow channel to facilitate the filling of a part.

Mold, runnerless injection molding A runner system in the mold that maintains the plastic resin in a molten state; therefore, no runners are ejected with the molded part, (see *Mold runner, hot, for thermoplastic*) or for thermoset plastic (see *Mold runner, cold, for thermoset*). See *also Mold, sprueless.*

Mold runner, unbalanced A runner in a multicavity mold where the distances from the sprue to the cavities are different.

Mold seam A line formed by a mold component such as removable members in a cavity, cam slides, etc. The prominence of the line depends on the accuracy with which the mating parts are matched. Usually, the line formed by the mold halves is called the mold parting line.

Mold, semipositive As the two halves of a semipositive thermoset compression mold begin to close, the mold acts much like a flash mold. The excess material is allowed to escape around the loose-fitted plunger and cavity. As the plunger telescopes further into the cavity, the mold becomes a positive mold with very little clearance, and full pressure is exerted on the material, producing a part of maximum density. This type of mold combines to advantage the free flow of material in a flash mold and the capability of producing dense parts in a positive mold.

Mold shrinkage Not the shrinkage of the mold, but the shrinkage that a molded part undergoes when it is removed from the mold and cooled to room temperature ("molded-

part shrinkage" would be a more appropriate phrase).

Mold, Siamese blow A colloquial term applied to the technique of blow molding two or more parts of a product in a single blow and then cutting them apart. Multiple cavities are used.

Mold side bar A loose piece used to carry one or more molding pins and operated from outside the mold.

Mold side coring Any of the projections used to core a hole in a direction other than the line of closing of a mold, and that must be withdrawn before the part is ejected from the mold. Also called side draw pin and side action mold. Figure 4-81 shows a cam pin action; it could have other mechanisms, such as a hydraulic cylinder. This figure also shows a puller pin for producing a sprue with a hollow section.

Mold spacer, insulating An insulator sheet placed between mold and platens to restrict heat transfer from mold to platens.

Mold spacer, parallel (1) One of the parallel support spacers placed between the mold and press plate or clamping plate. Also called risers. They take up space to allow a short mold to meet the machine minimum daylight opening. (2) A spacer placed between the hot plate and press platen to prevent the middle section of the mold from bending under pressure. (3) A pressure pad between the hot plates of a mold to control height when closed and to prevent crushing the parts of the mold when land area is inadequate.

Mold spherical diameter In a rotational molding process, the distance from the center of rotation, along a straight line at 45° to the vertical or horizontal, to the extremities of the mold swing.

Mold, split-ring A mold in which a split cavity block is assembled in a chase to permit the forming of undercuts in a molded piece. These parts are ejected from the mold and then separated from the molded piece. Also called split mold.

Mold, spring box A type of compression mold equipped with a spacing fork that prevents the loss of bottom-loaded inserts or fine details and that is removed after partial compression.

Mold sprue A tapered orifice in an injection or transfer mold through which plastic melt flows from the nozzle to the parting line, molded piece, or runner. The name sprue is also used for the plastic formed in this orifice.

Mold sprue bushing A hardened steel insert in an injection mold that contains the tapered sprue hole and has a suitable seat for the nozzle of the injection cylinder. It is sometimes called an adapter.

Mold sprue bushing, heated A mold element that contains a heating element to keep the plastic melt hot within the bushing. The bushing is inserted into the mold to provide a hot channel between the molding machine's nozzle and the mold cavity. Like the nozzle thermocouple temperature profile, the temperature profile in a heated sprue must be controlled for accurate closed-loop temperature control of thermoplastics and thermosets, particularly the former.

Mold sprue ejector pin When the undercut occurs on the cavity block retainer plate, this pin is called the sprue ejector pin.

Mold sprue gate A passageway through which melt flows from the nozzle of a molding machine to the cavity.

Mold, sprueless A mold in which the sprue-and-runner system is insulated from the mold.

Mold sprue lock or puller In injection molding thermoplastics, this is the portion of the melt that is held in the cold slug well by an undercut; it is used to pull the sprue out of

the bushing as the mold is opened. The sprue lock itself is pushed out of the mold by an ejector pin. When the undercut occurs on the cavity block retainer plate, this pin is called the sprue ejector pin.

Mold sprue puller A slotted pin used to remove a sprue from a sprue bushing.

Mold, stack A two-level mold, two sets of cavities stacked one above the other, for molding more parts per cycle. It is also called a three-plate mold, since a third or intermediate movable plate is used to make possible center or offset gating of each cavity on the two levels.

Mold, stacked four-level A four-face stack mold, capable of molding parts on four levels.

Mold, steam plate A mounting plate for compression thermoset molds, cored for circulation of steam.

Mold stop A metal part inserted between mold halves to control the thickness of a press-molded part. Not a recommended practice, because the plastic will receive less pressure, which can result in lower density and voids in the part.

Mold strength requirements The forces involved in the molding operation are compressive; they are exerted by the clamping ram and the internal melt pressure.

Mold stripper plate A plate that strips a molded piece from core pins or force plugs. The stripper plate is set into operation by the opening of the mold.

Mold temperature The final mold temperature is usually determined by the gate size and the processor's desire to attain short cycles. The smaller the gate, the higher the melt temperature must be in order to get melt through the gate. Therefore, the mold temperature is usually set low to remove the heat quickly from the melt and thus achieve short cycles. However, the colder the mold is, the quicker the plastic that first contacts the mold sets up (hardens). To prevent a part from cooling before the injection cycle has been completed, higher injection speeds and pressures are generally used.

Mold thickness There are minimum and maximum heights (thicknesses) of a mold that can be accommodated by the clamp end. Of the two figures, the maximum is more critical in case there is not sufficient machine clamping daylight opening. If the mold minimum thickness is less than the minimum daylight opening, spacer blocks are used to make up the difference.

Mold thread plug, ring, or core A part of a split mold that shapes a thread and must be unscrewed from the finished piece. Its purpose is to eliminate parting lines across the threads.

Mold types The mold is identified descriptively by a combination of terms such as the following: injection molding, compression molding, blow molding, reaction injection molding, rotational molding, and mold construction.

Mold unit A mold designed for quick changing of interchangeable cavity parts.

Mold variables Variations in mold conditions have a direct effect on part quality. Important factors are: mold temperature, mold venting, mold closing speed, mold surface condition (wear or damage), and mold accessories (core slides, neck inserts, etc.). The production of defective parts can be greatly reduced by recognizing the interrelationships of process and material variations as well as properly analyzing problems to provide timely, accurate solutions.

Mold wiper In injection molding, a device that enters between the opened mold halves during the ejection cycle, engages the molded piece, and lifts or shoves it from the mold. The wiper movement is interlocked with the mold