INJECTION MOLDING HANDBOOK

THIRD EDITION

EDITED BY

DOMINICK V. ROSATO, P. E.

DONALD V. ROSATO, PH.D.

MARLENE G. ROSATO, P. E.



Kluwer Academic Publishers
Boston/Dordrecht/London

Spectrum Ex. 1014
IPR Petition - USP 11,536,632

Distributors for North, Central and South America:
Kluwer Academic Publishers
101 Philip Drive
Assinippi Park
Norwell, Massachusetts 02061 USA
Telephone (781) 871-6600
Fax (781) 871-6528
E-Mail <kluwer@wkap.com>

Distributors for all other countries:
Kluwer Academic Publishers Group
Distribution Centre
Post Office Box 322
3300 AH Dordrecht, THE NETHERLANDS
Telephone 31 78 6392 392
Fax 31 78 6546 474
E-Mail <orderdept@wkap.nl>



Electronic Services <http://www.wkap.nl>

Library of Congress Cataloging-in-Publication Data

Injection molding handbook / Dominick V. Rosato, Donald V. Rosato, Marlene G. Rosato. – 3rd ed.
p. cm.
ISBN 0-7923-8619-1
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 Kluwer Academic Publishers.

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, Kluwer Academic Publishers, 101 Philip Drive, Assinippi Park, Norwell, Massachusetts 02061

Printed on acid-free paper.

Printed in the United States of America

Spectrum Ex. 1014
IPR Petition - USP 11,536,632

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

Spectrum Ex. 1014
IPR Petition - USP 11,536,632

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

Spectrum Ex. 1014
IPR Petition - USP 11,536,632

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

Spectrum Ex. 1014
IPR Petition - USP 11,536,632

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

Spectrum Ex. 1014
IPR Petition - USP 11,536,632

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

Spectrum Ex. 1014
IPR Petition - USP 11,536,632

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

Spectrum Ex. 1014
IPR Petition - USP 11,536,632

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

Spectrum Ex. 1014
IPR Petition - USP 11,536,632

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;

Spectrum Ex. 1014
IPR Petition - USP 11,536,632

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

Spectrum Ex. 1014
IPR Petition - USP 11,536,632

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

Spectrum Ex. 1014
IPR Petition - USP 11,536,632

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

Spectrum Ex. 1014
IPR Petition - USP 11,536,632

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

Spectrum Ex. 1014
IPR Petition - USP 11,536,632

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

Spectrum Ex. 1014
IPR Petition - USP 11,536,632

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,

Spectrum Ex. 1014
IPR Petition - USP 11,536,632

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

Spectrum Ex. 1014
IPR Petition - USP 11,536,632

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

Spectrum Ex. 1014
IPR Petition - USP 11,536,632

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

Spectrum Ex. 1014
IPR Petition - USP 11,536,632

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

Spectrum Ex. 1014
IPR Petition - USP 11,536,632

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

Spectrum Ex. 1014
IPR Petition - USP 11,536,632

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

8

Design Features That Influence Product Performance

Overview

Different techniques or methodologies are used to analyze premature molded product failures. Throughout this book the design problems (with solutions) that could cause failures are discussed. This chapter concentrates on this subject by providing more details. A variety of auditing methods and computer software programs are used or developed by molders to provide an analysis of potential problems. Although the actual time and cost to design a product may represent less than 5% of the total time and cost, the influence on the performance and cost of the final fabricated molded product is enormous.

Avoiding product failures can depend, in part, on the ability to predict the performance of all types of plastic materials and their shapes. With available time, the usual approach of product laboratory and/or field testing provides useful and reliable performance data. Engineers and designers continue to develop sophisticated computer methods for calculating stresses in complex structures while using different materials (18, 126, 416).

Computational methods have replaced the oversimplified models of material behavior formerly relied on. However, for new and very complex product structures that are being designed to significantly reduce the volume of materials used and, in turn the product cost, computer analysis is conducted on prototypes already fabricated and undergoing testing. This computer approach can result in early and comprehensive analysis of the effects of conditions such as temperature, loading rate, environment, and material defects on nonstructural and/or structural reliability. The information is supported by stress–strain behavior collected in actual material evaluations (Chap. 12).

When required, the finite element analysis (FEA) method has greatly enhanced the capability of the structural analyst to calculate displacement and strain–stress values in complicated structures subjected to arbitrary loading conditions. In its fundamental form, the FEA technique is limited to static, linear elastic analysis. However, there are advanced FEA computer programs that can treat highly nonlinear dynamic problems efficiently. Important features of these

Spectrum Ex. 1014
-26-
IPR Petition - USP 11,536,632

programs include their ability to handle sliding interfaces between contacting bodies and to model elastic–plastic material properties. These program features have made possible the analysis of impact problems that in the past had to be handled with very approximate techniques. The improved precision of FEA methods have provided better direction in locating high stress areas. Nonetheless, final verification of load-carrying capability may still require actual testing of the fabricated product. (Chap. 9, Finite Element Techniques).

Audits

Many companies periodically, usually on a specific schedule, perform an internal audit of their operations. The purpose is to verify system conformance of the complete operation for producing products that meet molded product performance and cost requirements. One of the goals is to ensure that no product failures occur at the end of the fabricating line or in service. Different guidelines are used to conduct these audits.

Unlike traditional internal auditing, auditing in the plastics industry might use a variable analysis flowchart. The analysis focuses on narrow or small segments of an operation and identifies as many variables as possible that could influence the accuracy or reproducibility of the product. It then explores how each variable can be controlled. Finally, it decides if the variables will be controlled and targets people (designer, moldmaker, etc.) who will implement the solution.

Quality system regulation A popular auditing technique is called the quality system regulation (QSR). In the past it was called good manufacturing practice (GMP) and process validation (PV). It is of particular importance for the medical device industry (which uses an extensive amount of plastics) and also in other product industries where strict processing procedures must be followed to eliminate failures. It sets up an important procedure to ensure meeting zero defects.

The originator of QSR, the U.S. FDA (Food & Drug Administration), defined this program as one providing a very high degree of assurance that a specific process will consistently produce a product meeting its predetermined specifications and quality attributes. Elements of validation are product specification, processing equipment, and process revalidation and documentation. QSR focuses almost exclusively on production practices and requires very detailed manufacturing procedures and extremely detailed documentation. The major requirements of QSR are in the areas of design, management responsibility, purchasing, and servicing. QSR encompasses quality system requirements that apply to the entire life cycle of a product or device.

The methods for performing the internal audit generally mimic those of an FDA inspection concerning the QRS. The company auditor examines its different operations beginning with the design group and proceeding through the plant's facility (equipment, material quality assurance, storage for materials and equipment, inspectors, etc.). The audits includes observing the employees performing all the operations, compares their performance to the written requirements and responsibilities, and reviews other conditions such as reviewing batch and production records for errors. A thorough audit could take hours to over a week to complete. When an audit is complete, the findings are summarized in a written report and circulated to the appropriate people (management of those departments where corrective actions are required).

Computer Approaches

Design methodologies are used via computer-aided design (CAD) approaches. They involve factors such as mold filling simulation and stress analysis to illustrate the design for manufacture considerations. Critical performance requirements such as exposure to service temperatures and dimensional stability can be studied. With the molding of the popular short fiber (predominantly glass fibers)

Spectrum Ex. 1014
IPR Petition - USP 11,536,632

molding compounds, variations in fiber orientations close to and away from the gate, as well as those resulting from mold design changes, can be evaluated (Chap. 9).

For example, varying molecular orientation can lead to undesirable anisotropy and nonuniformity in the part. The moduli along the flow and cross-flow directions as well as other relevant properties can be studied. Factors such as those reviewed are interrelated (Chap. 5). They must be considered in the design stage to eliminate or at least significantly reduce failures (1, 13, 18, 199).

Design Features That Influence Performance

Sound judgement and experience are basic requirements. Experience can be gained by working with someone that has experience and keeping up to date on the complete technology of injection molding (which includes reading and understanding all that is in this book, particularly this chapter). Familiarity with plastic behavior and processing methods are necessary. One of the earliest steps in product design is establishing the configuration of the part that will form the basis on which performance (dimensions, strength, etc.) calculations will be made and selecting a suitable material to meet the performance requirements. During this phase certain design features have to be keep in mind to avoid problems such as degradation of material properties. Such features, called detractors or constrains, are primarily responsible for the unwanted internal stresses that can reduce the available stress level for load bearing purposes (120).

Other features, classified as precautionary measures, may influence the favorable performance of a part if they are properly incorporated. For example, something as simple as a stiffening rib is different for an injection molded or structural foamed part even when both parts use the same material. Familiarizing oneself with design constraints is a critical first step in eliminating product problems. Product failures can range from simply not meeting aesthetic appearance to incompatible assembled products not properly attached (such as a molded part attached to a metal with a different thermal expansion coefficients).

This book contains information that can be used to set up a checklist of plastic capabilities and constraints based on performance requirements. Although it may not be the designer's job (in certain plants), that person should have some idea where problems can develop (Chap. 4, Correcting Mold Filling Imbalances in Geometrically Balanced Runner Systems; Chap. 11, Plastic Material and Equipment Variables) (7).

Another important consideration concerns tolerances and shrinkages (Chap. 4, Mold Shrinkages and Tolerances; Chap. 5, Molding Tolerances; and Chap. 9; Tolerances and Dimensional Controls). Plastic shrinkage is often reported as numbers that account for variations in processing conditions and wall thicknesses. Simple material and solidification models using *PVT* diagrams can be used as guides to define an effective molding pressure that characterizes the shrinkage that occurs during solidification under time-varying mold cavity pressures (Chap. 7). This effective pressure combines the thermal diffusivity of the material, the wall thickness, and the time-varying cavity pressure into a single parameter that is related to shrinkage (Chap. 5, Molding Tolerances; Chap. 9, Tolerances and Dimensional Controls) (1, 13, 18, 135, 302, 305, 390, 614).

Plastic Product Failures

The outstanding performance of plastics over long periods of time have been demonstrated in innumerable applications in all markets since 1886. Yet some users of plastic products continue to encounter disappointing results when using these materials. Practically all incidences of malfunctions are caused by a lack of knowledge of the characteristics and potentials of the plastics, with a resultant misapplication, rather than from any shortcomings in the material or processing itself. In other words, one must use the correct design, the correct material, and the proper fabrication technique to meet

Spectrum Ex. 1014
IPR Petition - USP 11,536,632

cost–property efficiency. For example, an analysis of long-term loading requires one to evaluate the plastic's creep behavior at the temperatures the plastic will be subjected to in service. Creep data analyses have been used extensively since at least 1940 with excellent results (Chap. 5, Long-Term Behavior of Plastics: Creep) (13, 18, 390).

Unfortunately many lower cost materials and processing techniques are available and the general public might assume these to be equal to the more expensive parts that will not fail. Industry standards and specifications are constantly being developed and used for public awareness of inferior products. Unfortunately, they are usually issued after the problem or failure develops in service.

Design Failure Theory

In many cases, a product fails when the material begins to yield "plastically." In a few cases, one may tolerate a small dimensional change and permit a static load that exceeds the yield strength. Actual fracture at the ultimate strength of the material would then constitute failure. The criterion for failure may be based on normal or shear stress in either case. Fatigue failure is the most common mode of failure. Other modes of failure include excessive elastic deflection or buckling. The actual failure mechanism may be quite complicated; each failure theory is only an attempt to explain the failure mechanism for a given class of materials. In each case a safety factor is employed. However, with proper part design, these failures can be eliminated or can be permitted if part performance is met (18).

Basic Detractors and Constraints

The successful design and fabrication of good plastic products require a combination of sound judgment and experience. Designing good products requires a knowledge of plastics that includes their advantages and disadvantages as well as some familiarity with processing methods. Until the designer becomes familiar with processing, a fabricator must be taken into the designer's confidence early in development and consulted frequently during those early days. The fabricator and mold or die designer should advise the product designer on materials behavior and how to simplify processing. The designer should not become restricted by understanding only one process or method and in particular should avoid focusing on just a certain narrow aspect of a design or process. (Chap. 15).

Although there is no limit theoretically to the shapes that can be created, practical considerations must be met. These relate not only to part design but also to mold or die design, since these must be considered one entity in the total creation of a usuable, economically feasible part. In the sections that follow, various phases considered important in the creation of such parts are examined for their contribution to and effect on design and function.

Prior to designing a part, the designer should understand such basic factors as those summarized in Fig. 8-1. Success with plastics, or any other material, for that matter, is directly related to observing design details. For example, something as simple as a stiffening rib is different for an injection molded or structural foam part, even though both parts may be molded from the same plastic (see Fig. 8-1). And a stiffening rib that is to be molded in a low-mold-shrink, amorphous TP will differ from a high-mold-shrinkage, crystalline TP rib, even though both plastics are just injection-molded (see Chaps. 4 and 6). Ribs molded in RP/composite plastics have their own distinct requirements:

The important factors to consider in designing can be categorized as follows: part thickness, tolerance, ribs, bosses and studs, radii and fillets, draft or taper, holes, threads, color, surface finish and gloss level, decorating operations, the parting line, gate locations, molded part shrinkage, assembly techniques, mold or die design, production volume, the tooling and other equipment amortization period, as well as the plastic and machine selected. The order of importance that these factors follow can vary, depending on the product to be designed and the

Spectrum Ex. 1014
IPR Petition - USP 11,536,632

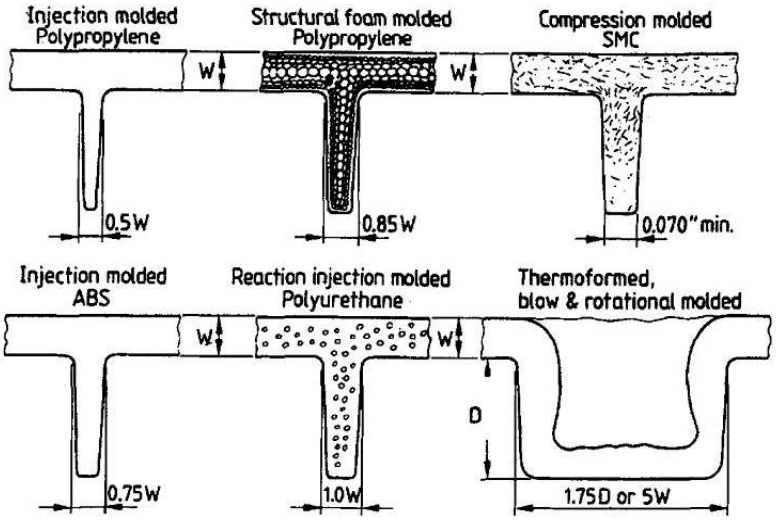


Fig. 8-1 Example of how different plastics and molding processes can affect the design details of a stiffening rib.

designer's familiarity with particular materials and processes (Table 8-1).

Preparing a complete list of design constraints is a crucial first step in plastic part design; failure to take this step can lead to costly errors. For example, a designer might have an expensive injection mold prepared, designed for a specific material's shrink value, only to discover belatedly that the initial material chosen did not meet some overlooked design constraint. Flammability, glass-fiber fillers to provide a higher modulus, and other requirements are best considered before a tool is made. Otherwise, the designer may

Table 8-1 Simplified example of the effect of rib and cross-section changes

Geometry	Cross-Sectional Area sq. in. (mm^2)	Maximum Stress, psi (MPa)	Maximum Deflection, in. (mm)
(19.1 mm) 0.75 ORIGINAL SECTION 0.080 (2.0)	0.0600 (38.7)	6,800 (46.9)	0.694 (17.6)
0.040 0.400 ORIGINAL SECTION WITH RIB	0.0615 (39.7)	2,258 (15.6)	0.026 (0.66)
0.75 THICK SECTION 0.239	0.1793 (115.7)	2,258 (15.6)	0.026 (0.66)

Spectrum Ex. 1014
IPR Petition - USP 11,536,632

have the difficult if not impossible task of finding a plastic that does meet all the design constraints, including the important appropriate shrink value for the existing mold. Such desperation in the last stages of a design project can and should be avoided. As emphasized from one end of this book to the other, it is vital to set up a complete checklist of product requirements, to preclude the possibility that a critical requirement may be overlooked initially. Recognize that the "impossible" as well as the "approaches" to be avoided can in most cases still produce excellent products; however, it is easier to follow the direction with the least number of problems.

Tolerance and Shrinkage

Two different forms of shrinkage must be considered when designing to meet tolerances: the initial shrinkage that occurs while a part is cooling, called the mold or die shrinkage, and that which occurs after as many as 24 hr, called the after-shrinkage or afterswell. Some plastics are more stable than others after aging, regardless of their initial shrinkage. In many cases, low shrinkage may indicate greater stability. Some plastics have zero shrinkage, with others exhibiting little or a high degree of shrinkage.

Large, unpredictable shrinkages can make close-tolerance designing almost impossible, so these must be indicated on the drawings. If it has been determined in advance that a part must be postcured, stress-relieved, or baked, allowance must be made for probable additional shrinkage. These requirements must be specified on the drawings.

Especially for long runs, mold or die design is an important factor. The metals that will be used, particularly in mold cavities, and the forces required will largely be determined by the complexity of the product design. This complexity will, of course, dictate in turn the intricacy of the tool design that will eventually be used. In general, pack hardening, oil hardening, and prehardened steels are used, with materials such as beryllium copper and electroformed cavities finding use in applications for specialized purposes. Chrome plating is frequently used to protect parts from the corrosive effect of volatiles present in some materials. Plating not only produces higher luster and prevents tool staining but also eliminates the sticking of parts on removal from a mold or die. However, plating will only duplicate an existing surface, so the tool itself must be highly polished before being plated. The intricacy required in tool design, and the commensurate costs, may be leading factors when choosing a plastic.

If the material must flow around pins, spiders, and projections, it must have suitable flow properties and weld properly, leaving only minimal flow marks (see Chap. 4). Thus, the availability of the proper material becomes an important consideration (Table 8-2).

Once shrinkage and postmolding shrinkage phenomena are known, the dimensional changes occurring in parts molded from a given plastic during the initial shrinkage (which takes place in the mold cavity as the melt solidifies) can be anticipated with reasonable accuracy. As previously explained, further contraction (postmolding shrinkage) may still occur in molded parts as internal stresses are relieved. Other opposite changes (expansions) can be caused by humidity absorbed after molding.

The molding shrinkage of thermosets (TSs) and amorphous thermoplastics (TPs) such as PS, CA, PMMA, PC, etc. is generally below 1% and no appreciable postmolding shrinkage occurs. With partly crystalline structure TPs such as PE, PP, PA, POM, etc., molding shrinkage may range from 1 to 4% depending on the type of plastic and presence of inert fillers (glass, mica, $CaCO_3$, etc.). Such plastics also exhibit varying degrees of postmolding shrinkage, with TSs usually having the least. These types of considerations show that accurately determining the extent to which mold cavity dimensions should be increased to compensate for shrinkage is not an easy task when one has no experience in this area of technology. As reviewed throughout this book, molding shrinkage depends not only on the plastic's intrinsic properties but also on many variables such as

Spectrum Ex. 1014
IPR Petition - USP 11,536,632

Table 8-2 Guidelines for nominal TP mold-shrinkage rates using ASTM $\frac{1}{4}$- and $\frac{1}{2}$-in.-thick test specimens

Material	Average Rate[a] per ASTM D 955	
	0.125 in. (3.18 mm)	0.250 in. (6.35 mm)
ABS		
Unreinforced	0.004	0.007
30% glass fiber	0.001	0.0015
Acetal, copolymer		
Unreinforced	0.017	0.021
30% glass fiber	0.003	NA
HDPE, homo		
Unreinforced	0.015	0.030
30% glass fiber	0.003	0.004
Nylon 6		
Unreinforced	0.013	0.016
30% glass fiber	0.0035	0.0045
Nylon 6/6		
Unreinforced	0.016	0.022
15% glass fiber + 25% mineral	0.006	0.008
15% glass fiber + 25% beads	0.006	0.008
30% glass fiber	0.005	0.0055
PBT polyester		
Unreinforced	0.012	0.018
30% glass fiber	0.003	0.0045
Polycarbonate		
Unreinforced	0.005	0.007
10% glass fiber	0.003	0.004
30% glass fiber	0.001	0.002
Polyether sulfone		
Unreinforced	0.006	0.007
30% glass fiber	0.002	0.003
Polyether-etherketone		
Unreinforced	0.011	0.013
30% glass fiber	0.002	0.003
Polyetherimide		
Unreinforced	0.005	0.007
30% glass fiber	0.002	0.004
Polyphenylene oxide/PS alloy		
Unreinforced	0.005	0.008
30% glass fiber	0.001	0.002
Polyphenylene sulfide		
Unreinforced	0.011	0.004
40% glass fiber	0.002	NA
Polypropylene, homo		
Unreinforced	0.015	0.025
30% glass fiber	0.0035	0.004
Polystyrene		
Unreinforced	0.004	0.006
30% glass fiber	0.0005	0.001

[a] Rates in in./in. (Courtesy of ICI-LNP.)

Spectrum Ex. 1014
IPR Petition - USP 11,536,632

part shape, mold design, and molding process (7.18).

From a knowledge of these factors affecting mold shrinkage, a mold designer should be able to anticipate fairly accurately the shrinkage a given part will undergo and to avoid major miscalculations that would entail costly mold remakes or alterations. This is a highly demanding task that moldmakers have been consistently performing.

There is an SPI standard that specifies guide limits for certain dimensions, and each material supplier converts the data to suit a specific plastic. In Chap. 4, Tables. 4-13 and 4-14 give examples of such information. This type of information is intended to give the designer a guide for tolerances that are to be shown on drawings; these tolerances include variation in part manufacture and some variation in tooling.

Tolerances on dimensions should be specified only where absolutely necessary. Too many drawings show limits of sizes when other means of attaining desired results would be more constructive. For example, if the outside dimensions of drill housing halves were to have a tolerance of ±0.003 in. (0.0076 cm), this would be a tight limit. Yet if half of the housing were to be on the minimum side and the other on the maximum side, there would be a resulting step that would be uncomfortable to the feel of the hand while gripping the drill. A realistic specification would call for matching of halves that would provide a smooth joint between them, with the highest step not to exceed 0.002 in. (0.0051 cm). The point is that limits should be specified in such a way that those responsible for the manufacture of a product will understand the goal that is to be attained. Thus, we may indicate "dimensions for gear centers," "holes as bearing openings for shafts," "guides for cams," etc. This type of designation would alert a moldmaker as well as a molder to the significance of the tolerances in some areas and the need for matching parts in other places and clearance for assembly in still other locations.

Most of the engineering plastics faithfully reproduce the mold configuration, and when processing parameters are appropriately controlled, they will repeat with excellent accuracy.

Plastic gears and other precision parts made of acetal, nylon, polycarbonate, etc. have tooth contour and other precision areas made with a limit of 0.002 in. (0.0051 cm), and the spacing of the teeth is uniform to meet the most exacting requirements.

The problem with any precise part is to recognize what steps are needed to reach the objective and to follow through every phase of the process in a thorough manner to safeguard the end-product. Throughout this book "shrinkage" is reviewed based on material and processing characteristics. Different factors can cause variation in shrinkage; indeed, the way processing parameters can influence dimensional variation is very important. Some materials perform better than others in that respect.

Generally, if we approach tolerances according to their purposes [(1) functional requirements, such as running it, sliding fit, gear tooth contour, etc.; (2) assembly requirements—that is, to accommodate parts with their own tolerances; and (3) matching parts for appearance or utility], we should come up with feasible tolerances that will be reasonable and useful. This will be more productive than trying to apply tolerances strictly on a dimensional basis.

Tolerances should be indicated only when they are needed, carefully analyzed for their magnitude, and of proven usefulness.

The adaption of metal tolerances to plastics is not advisable. The reaction of plastics to moisture and heat, for example, is drastically different from that of metals, so that pilot testing under extreme use conditions is almost mandatory for establishing adequate tolerance requirements.

Using these types of calculated shrinkage, theory can dictate how oversized to cut the tool (the mold or die) if a part has a relatively simple shape. For other shapes some critical key dimensions of the part will, more often than not, not be as predicted from the shrink allowance, particularly if the item is long, complex, or tightly toleranced. The important factors that influence the shrinkage of a specific plastic in using a specific machine,

Spectrum Ex. 1014
IPR Petition - USP 11,536,632

Table 8-3 Functions of an injection mold

Mold Component	Function Performed
Mold base	Hold cavity (cavities) in fixed, correct position relative to machine nozzle.
Guide pins	Maintain proper alignment of the two halves of a 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 retraced position as mold closes for next cycle.

such as injection molding, by causing it to vary and not follow the expected values are flow direction, wall thickness, flow distance, and the presence of reinforcing fibers.

Determining shrinkage involves more than just applying the appropriate correction factor from a material's data sheet. Shrinkage is caused by a volumetric change in a plastic as it cools from a molten to solid form. As discussed in Chaps. 4 and 6, shrinkage is not a single event. Additional shrinkage can occur when frozen-in stress is relieved by annealing or exposure to high service temperatures.

The main considerations in mold design affecting part shrinkage are to provide adequate cooling, proper gate size and location, and structural rigidity. Of the three, cooling conditions is the most critical, especially for crystalline resins (see Chap. 6). The cooling system must be adequate for the heat load. Slow cooling increases shrinkage by giving resin molecules more time to reach a relaxed state. In crystalline types, longer cooling times lead to a higher level of crystallinity, which in turn accentuates shrinkage. Proper cooling, along with having an overall melt-flow analysis of how the material will react in the mold, prepared by the mold designer, will eliminate or at least be capable of controlling the potential problems of shrinkage and warpage. This analysis can also include the best gate locations. Table 8-3 summarizes functions of an injection mold.

Several computer-aided flow-simulation programs now offer modules designed to forecast part shrinkage—and, to a limited degree, warpage—from the interplay of resin and mold temperatures, cavity pressures, stress, and other variables in mold-fill analysis. The predicted shrinkage values in various areas of the part should be used as the basis for sizing the mold cavity, either by manual input or feed-through to the mold-dimensioning program. All the programs can successfully predict a certain amount of shrinkage under specific conditions.

Shrinkage and warpage Potential warping of a part is always a major concern in mold design. Warping is caused by differential shrinkage, that is, if one area of the molding has a different level of shrinkage from another area, the part will warp. Another cause is if one area is overpacked, while another area of the mold has much less melt pack. The classic example of warpage due to orientation occurs with centrally gated parts (Fig. 8-2).

If the cooling around the sprue takes longer than around the edge, then differential

Spectrum Ex. 1014
IPR Petition - USP 11,536,632

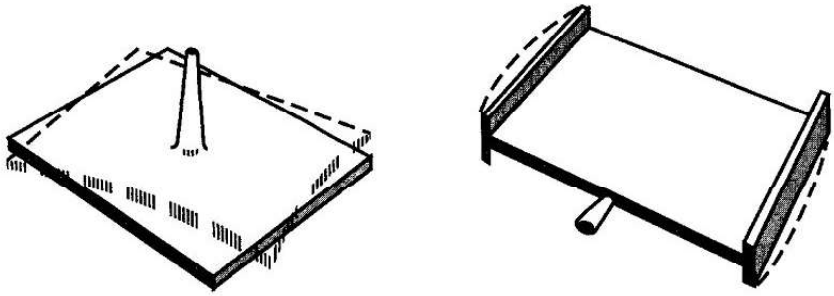


Fig. 8-2 Shrinkage across the diagonal is higher than along the edges, so there is tension force along the diagonal causing the corners to curl up or down.

cooling will cause the part to warp. This variation in cooling times could be caused either by frictional heating directly or improper cooling design. There is much more heat to be extracted from the gate area than the edge; therefore, cooling channels must be designed to extract more heat from the gate. Twin cooling circuits could be used for better local temperature control.

The cooling of both the core and cavity must be carefully planned. It is very easy to cool the cavity, but it is quite difficult to obtain good cooling of the core, particularly in the corners. If the corners of the core are hotter than the cavity, then differential cooling will result, deflecting the side inward.

If the material is crystalline, the thickness around the rim can be increased, thus increasing the shrinkage on the edge. This will offset the difficulty of cooling around the gate. Although this is very workable, a check must be made to ensure that a racetrack effect has not been created.

A clear understanding of the cause of warpage of injection-molded parts has led to practical shrinkage and warpage software. Warpage is caused by variations in shrinkage throughout the part (a few examples have been discussed). These variations may appear from region to region, through the thickness, and as different shrinkages in directions parallel and perpendicular to the plastic orientation direction.

Flow and cooling analysis results are used to predict shrinkage strains, taking the above variations into account. These strains are then input to a structural analysis program to find the deformation of the component. The software may be used to indicate which of the three variations has the most effect on warpage, which is very useful information for a product designer.

The traditional approach to shrinkage prediction is to correlate against molding variables such as thickness, mold temperature, etc. This approach is tied to a particular mold geometry. For a general shrinkage prediction, it is important to correlate shrinkage against more fundamental variables (152). These variables are volumetric shrinkage (from *PVT* data, Chap. 7), crystalline content, mold restraint effects, and orientation effects. These variables are derived from a finite-element/finite-difference filling and packing analysis, using *PVT* data to model compressibility and thermal contraction effects.

The flow analysis uses mold temperature as a boundary condition, so initially a constant value is set. The plastic temperature results from the flow analysis are then used in a cooling analysis to compute the end-of-cycle mold temperature at each finite element. This detailed mold temperature information is then used in a second flow analysis, to produce the final grid-point results that are used in shrinkage calculations. A careful account of these variations leads to a practical tool for the prediction and minimization of warpage.

Residual Stress

Residual stresses can cause warpage. Processing-induced stresses that influence

Spectrum Ex. 1014
IPR Petition - USP 11,536,632

properties such as mechanical, physical, environmental, and aesthetic factors (which also exist in metals and ceramics) can have favorable or unfavorable effects, depending on the application of the load with respect to the direction of the stresses or orientation. For example, at room temperature an unoriented PS is a brittle, glassy, amorphous polymer, whereas a uniaxial oriented PS is highly anisotropic. High tensile strength, elongation, and resistance to environmental stress crazing and cracking are achieved in the direction of orientation. However, an oriented PS is weaker and more susceptible to stress crazing in its transverse direction than an unoriented PS. A biaxially oriented PS is strong and tough in all directions (7, 18).

Residual stresses and molecular orientation play an important role in the toughness enhancement of cold-worked plastics, because toughness is primarily based on the mechanics of craze formation and shear band (crazes and flaws) formation. The shear bands determine the fracture mode and toughness of a polymer when subjected to impact loads. The amount of energy dissipated will depend on whether the material surrounding the flaws deforms plastically. For toughness enhancement the residual stresses play an important role in the suppression of craze formation, by avoiding the stress state that promotes brittle fracture.

The term *residual stresses* identifies the system of stresses that are, in effect, locked into a part, even without external forces acting on it. For instance, minute stresses may be induced in a material by nonuniform heating and cooling. The production of residual stresses is usually the result of non-homogeneous plastic deformation occurring during thermal and mechanical actions, arising from changes in either volume or shape. Thermal treatments such as quenching (rapid cooling) and annealing (slow cooling) introduce changes in physical and mechanical properties. For example, with sheet plastic the stresses created by quenching are the result of uneven cooling, when the surfaces cool faster than the core. This produces nonuniform volume changes and properties throughout the thickness. The compressive stresses on the surface of the quenched plastic produce tensile stresses in the core, which maintain the equilibrium of the forces.

Stress Concentration

Sharp corners should always be avoided in designing, particularly when working with TP injection-molded parts. Although sharp-cornered designs are common with certain sheet metal and machined parts, good design practice in any material dictates the use of generous radii, to reduce stress concentrations. Reinforced plastics and composites and metal parts will often tolerate sharp corners, because the stresses at their corners are low compared to the strength of the material or localized yielding redistributes the load. However, neither of these factors should be relied on in TP-molded parts. Sharp corners, particularly the inside corners, can cause severe molded-in stresses as a material shrinks onto the corner, as well as poor flow patterns, reduced mechanical properties, and increased tool wear.

The elementary formulas used in design are based on structural members having a more or less constant cross section, or at least only a gradual change of contour, but these conditions are seldom found in practice. The presence of shoulders, bosses, grooves, holes, threads, and corners results in modifying the simple stress distribution and in localized, higher stresses. This localization, known as the stress concentration factor, is defined as K = maximum stress divided by nominal stress. Localized high stresses must in most cases be determined experimentally rather than theoretically. The photoelastic technique is one of the more effective methods used to do this. To interpret a photoelastic diagram qualitatively, it is sufficient to know that the number of fringes (the density of lines) is proportional to the absolute stress level.

In the vicinity of a sharp corner all fringes converge toward the apex. Having a high density of lines at this point indicates the presence of a high stress level. At a rounded corner, there will be considerably

Spectrum Ex. 1014
IPR Petition - USP 11,536,632

less concentration. Besides molding problems, sharp corners often cause premature failure because of the stress concentration. To avoid these problems, inside corner radii should be equal to one-half the nominal wall thickness, with a 0.020-in. radius considered a minimum for parts subjected to stress and a 0.005-in. minimum for the stress-free regions of a part. Having inside radii less than 0.005 in. is not recommended for most materials. Outside corners should have a radius equal to the inside corner plus the wall's thickness.

Figure 8-3 illustrates quantitatively the effect of fillet radius (R) on stress concentration factor K. Assume that a force or load is being exerted on the cantilevered section shown. As the radius is increased, with all other dimensions remaining constant, R/T increases proportionally and the stress-concentration factor decreases, as shown by the curve. The K factor can be reduced by 50%, from 3.0 to 1.5, by increasing the R/T fivefold, from 0.1 to 0.5. This curve shows how readily the K factor can be reduced by using a large fillet radius. A fillet with an optimal design is obtained with an R/T of 0.5. A further increase in the radius reduces the stress concentration by only a marginal amount. Stress concentration factors on the same order of magnitude have also been determined for grooves, notches, holes, screw threads, bosses, and ribs.

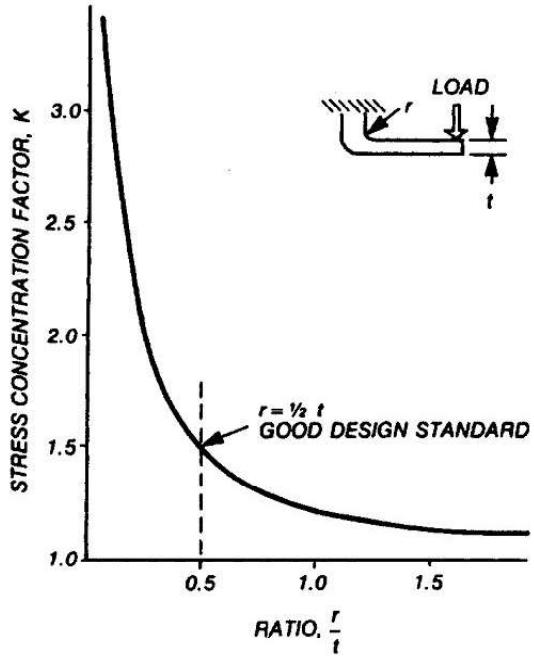


Fig. 8-3 Effect of the fillet radius on the stress concentration factor. To find the stress associated with having a small radius, multiply the calculated bending stress by K.

Sink Mark

Sink marks are indentations on the surface of a molded part that can occur usually when there is a significant local change in wall thickness. Examples include ribs, bosses, and undercuts. They are caused by thermal contraction of the melt during cooling in the mold. Since the volumetric change of plastic from melt to the solid could be about 25% and the compressibility of plastics occurs at a lower amount such as 15%, it is possible to pack out a mold. This action could produce sink marks in the pressurization phase. Some compensating flow is necessary to eliminate the sink marks. It is impossible to eliminate or minimize sink marks by adjusting holding pressure from high to low as a rule of thumb.

Analyzing flow as a combination of viscous fluid flow and heat transfer can aid in understanding what is happening in the mold (Chap. 7). The object is to flow plastic through the thin section 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 situation the plastic could freeze off. With a high holding pressure, there would 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 section would not remain molten long enough for the thick section such as a boss to be adequately packed out.

Design Concept

In designing a totally new product or redesigning an existing one to improve the product, bring about cost savings, or some combination of these or other reasons, consideration should be given to the key advantages of injection molding. These advantages include the ability to produce finished,

Spectrum Ex. 1014
IPR Petition - USP 11,536,632

multifunctional, or complex molded parts accurately and repeatedly in a single, highly automated operation (Chaps. 2, 5, and 7). While keeping this in mind during the initial planning stage, one should also be aware of the general design considerations presented in this section.

Many parts of an injection mold will influence the final product's performance, dimensions, and other characteristics. These mold parts include the cavity shape, gating, parting line, vents, undercuts, ribs, hinges, etc. The mold designer must take all these factors into account. At times, to provide the best design the product designer, processor, and mold designer may want to jointly review where compromises can be made to simplify meeting product requirements. With all this interaction, it should be clear why it takes a certain amount of time to ready a mold for production (7, 18).

Thus, in the design of any injection-molded part, there are certain desirable goals that the designer should use. In meeting them, problems can unfortunately develop. For example, the most common mold design errors of a sort that can be eliminated usually occur in the following areas:

Thick or thin sections, transitions, warp, and stress

Multiple gates and weld lines

Wrong gate locations

Inadequate provision for cavity air venting

Parts too thin (such as diaphragms) to mold properly,

Parts too thick to mold properly

Plastic flow path too long and tortuous

Runners too small

Gates too small

Poor temperature control

Runner too long

Part symmetry vs. gate symmetry

Orientation of polymer melt in flow direction

Hiding gate stubs

Stress relief for interference fits

Living hinges

Slender handles and bails

Thread inserts

Creep or fatigue over long-time stress (extremely important)

As seen in previous chapters, different plastics have different melt and flow characteristics. What is used in a mold design for a specific material may thus require a completely different type of mold for another material. These two materials might, for instance, have the same polymer but use different proportions of additives and reinforcements. This situation is no different than that of other materials such as steel, wood, ceramics, and aluminum.

It is important to recognize that the drawing of a plastic product will not specifically spell out the way many of its details will be carried out in the mold design. Some features adversely affect the strength and quality of the molded product. In most cases, these problem details can be modified by the designer to minimize their adverse effects on the properties of the part. What follows is a general summary of how to reduce problems to tolerable limits.

First, inside sharp corners should normally be shown as two intersecting straight lines, without specific indication as to the functional requirement or degree of sharpness. Inside square corners are areas of stress concentration, quite similar to a notch in a test bar. The Izod impact strength of notched and unnotched test bars shows the relative impact strength of each material at the two conditions.

Thus, for example, polycarbonate has an impact strength of the notched $\frac{1}{8}$-in. test bar of 12 to 16 ft-lb/in. of notch, whereas the same bar unnotched does not fail the test. Polypropylene has an impact strength 30 times greater in the unnotched than notched bar. Nylon shows a drastic increase in impact strength as the radius increases from sharpness to $\frac{3}{64}$ R. A similar trend exists for most other materials. These examples point out that brittleness increases with the decreasing of a radius in a corner. Visually, a radius

Spectrum Ex. 1014
IPR Petition - USP 11,536,632

of 0.020 in. on a plastic part may be considered sharp, with an influence on strength that is much more favorable than a radius of 0.004 in. To the moldmaker, a sharp corner is usually easier to produce, but in the plastic part it is a source of brittleness and, in most cases, highly undesirable. Inside sharp corners frequently appear on plastic-part drawings. It is the mold designer's responsibility to call attention to such strength degradation and invite appropriate corrective measures.

Second, varying wall thicknesses from thick and thin sections in a part will lead to problems in molding. Having a uniform wall throughout a part gives it good strength and appearance. Thick and thin sections will have molded-in stresses, different rates of shrinkage (causing warpage), and possibly void formation in the thick portion. Since the parts in a mold solidify from their outer surfaces toward the center, sinks will tend to form on the surface of a thick portion. When thick ($\frac{3}{16}$ in. and over) and thin ($\frac{1}{8}$ in. or less) portions are unavoidable, the transition should be gradual and coring should be utilized whenever possible.

Third, sinks are not only the result of the causes listed above; they also occur whenever supporting or reinforcing ribs, flanges, or similar features are used in an attempt to provide functional service without changing the basic wall thickness of a product. If the appearance of a sink on the surface is objectionable, the ribs and transition radius should be proportioned so that their contribution to the sink is minimal. Figure 8-4 is a guide to the dimensioning of ribs.

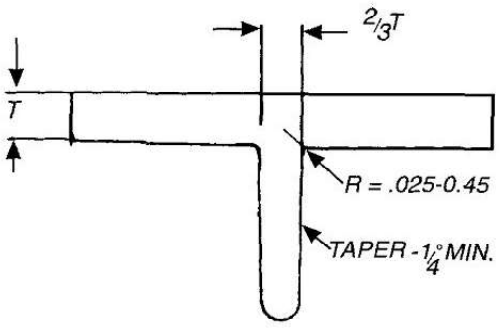


Fig. 8-4 Rib and wall dimensioning.

Fourth, molded-in metal parts should be avoided whenever alternate methods will accomplish the desired objectives. If incorporating such inserts is essential, they should be shaped so that they will present no sharp inside corners to the plastic. The effect of the sharp edges of a metal insert would be the same as explained in the first point above, namely, brittleness and stress concentration. The cross section that surrounds a metal insert should be heavy enough to avoid cracking upon cooling. A method of minimizing cracking around the insert is to heat the metal insert prior to mold insertion to a temperature of 250 to 300°F so that it will tend to thermoform the plastic into its finished shape. The thickness of the plastic enclosure will vary from material to material. A reasonable guide is to have the thickness 1.75 to 2 times the size of the insert diameter (Table 8-4).

Fifth, most plastic parts are used in conjunction with other materials. If the in service temperature is other than room temperature, certain compensatory steps must be taken to avoid problems arising from the difference in the thermal coefficients of expansion of the different materials. Many plastic materials expand about 10 times as much as steel. Thus, careful analysis of the conditions under which the metallic materials are coemployed for functional uses is called for. In the automotive industry, many long plastic parts are used in conjunction with metal frames. If proper compensation is not made for the difference in thermal coefficients of the materials, buckling or looseness may result, causing noise or a poor appearance.

Finally, plastic threads have a very limited strength and may be further degraded if the thread form is not properly shaped. The V-shaped portion at the outside of a female thread will present a sharp inside corner that will act as a stress concentrator and thereby weaken the threaded cross section. A rounded form that can be readily incorporated in a molding insert will appreciably improve the strength over a V-shaped form. When self-tapping, thread-cutting, or thread-form screws are used, their holding power can be increased if either the screws or plastic

Spectrum Ex. 1014
IPR Petition - USP 11,536,632

Table 8-4 Suggested minimum wall thicknesses for inserts of various diameters

Plastic Material	Diameter of Inserts, in. (mm)					
	0.125 (3.17)	0.250 (6.35)	0.375 (9.52)	0.500 (12.7)	0.750 (19.0)	1.00 (25.4)
ABS	0.125 (3.17)	0.250 (6.35)	0.375 (9.52)	0.500 (12.7)	0.750 (19.0)	1.00 (25.4)
Acetal	0.062 (1.57)	0.125 (3.17)	0.187 (4.75)	0.250 (6.35)	0.375 (9.52)	0.500 (12.7)
Acrylics	0.093 (2.36)	0.125 (3.17)	0.187 (4.75)	0.250 (6.35)	0.375 (9.52)	0.500 (12.7)
Cellulosics	0.125 (3.17)	0.250 (6.35)	0.375 (9.52)	0.500 (12.7)	0.750 (19.0)	1.00 (25.4)
Ethylene vinyl acetate	0.040 (1.02)	0.085 (2.16)	N.R.	N.R.	N.R.	N.R.
FEP (fluorocarbon)	0.025 (0.64)	0.060 (1.52)	N.R.	N.R.	N.R.	N.R.
Nylon	0.125 (3.17)	0.250 (6.35)	0.375 (9.52)	0.500 (12.7)	0.750 (19.0)	1.00 (25.4)
Noryl (modified PPO)	0.062 (1.57)	0.125 (3.17)	0.187 (4.75)	0.250 (6.35)	0.375 (9.52)	0.500 (12.7)
Polyallomers	0.125 (3.17)	0.250 (6.35)	0.375 (9.52)	0.500 (12.7)	0.750 (19.0)	1.00 (25.4)
Polycarbonate	0.062 (1.57)	0.125 (3.17)	0.187 (4.75)	0.250 (6.35)	0.375 (9.52)	0.500 (12.7)
Polyethylene (HD)	0.125 (3.17)	0.250 (6.35)	0.375 (9.52)	0.500 (12.7)	0.750 (19.0)	1.00 (25.4)
Polypropylene	0.125 (3.17)	0.250 (6.35)	0.375 (9.52)	0.500 (12.7)	0.750 (19.0)	1.00 (25.4)
Polystyrene	Not recommended					
Polysulfone	Not recommended					
Surlyn (ionomer)	0.062 (1.57)	0.093 (2.36)	0.125 (3.17)	0.187 (4.75)	0.250 (6.35)	0.312 (7.92)
Phenolic GP	0.093 (2.36)	0.156 (3.96)	0.187 (4.75)	0.218 (5.53)	0.312 (7.92)	0.343 (8.71)
Phenolic (medium-impact)	0.078 (1.98)	0.140 (3.56)	0.156 (3.96)	0.203 (5.16)	0.281 (7.14)	0.312 (7.92)
Phenolic (high-impact)	0.062 (1.57)	0.125 (3.17)	0.140 (3.56)	0.187 (4.75)	0.250 (6.35)	0.281 (7.13)
Urea	0.093 (2.36)	0.156 (3.96)	0.187 (4.75)	0.218 (5.53)	0.312 (7.92)	0.343 (8.71)
Melamine	0.125 (3.17)	0.187 (4.75)	0.218 (5.53)	0.312 (7.92)	0.343 (8.71)	0.375 (9.52)
Epoxy	0.020 (0.51)	0.030 (0.76)	0.040 (1.02)	0.050 (1.27)	0.060 (1.52)	0.070 (1.78)
Alkyd	0.125 (3.17)	0.187 (4.75)	0.187 (4.75)	0.312 (7.92)	0.343 (8.71)	0.375 (9.52)
Diallyl phthalate	0.125 (3.17)	0.187 (4.75)	0.250 (6.35)	0.312 (7.92)	0.343 (8.71)	0.375 (9.52)
Polyester (premix)	0.093 (2.36)	0.125 (3.17)	0.140 (3.56)	0.187 (4.75)	0.250 (6.35)	0.281 (7.14)
Polyester TP	0.062 (1.57)	0.125 (3.17)	0.187 (4.75)	0.250 (6.35)	0.375 (9.52)	0.375 (9.52)

parts are heated to a temperature of 180 to 220°F at joining time. This will provide thermoforming action to some degree and keep the stress level caused by the joining action at a low point.

These possible sources of problems in a molded part should be marked on the part drawing and explained to the product designer for corrective action or creating an awareness of possible product defects from design limitations. This is a necessary step in the chain of events in which the aim is to produce a tool that will provide parts for a good working product. Even if the mold's design, workmanship, and operation are carried out to the highest degree of quality, they cannot overcome a built-in weakness of product design.

Terminology

Sharp Corners

When a part drawing does not show a radius, the tendency is for the toolmaker (while making a mold) to leave the intersecting machined or ground surfaces as they are generated by the machine tool. The result is a sharp corner on the molded part. Such sharp

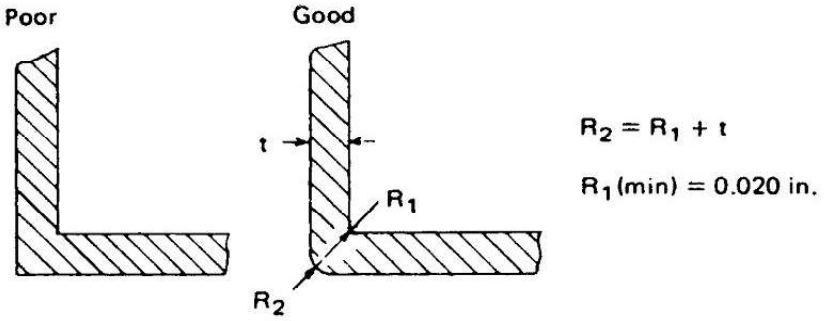


Fig. 8-5 An inside corner.

Spectrum Ex. 1014
IPR Petition - USP 11,536,632

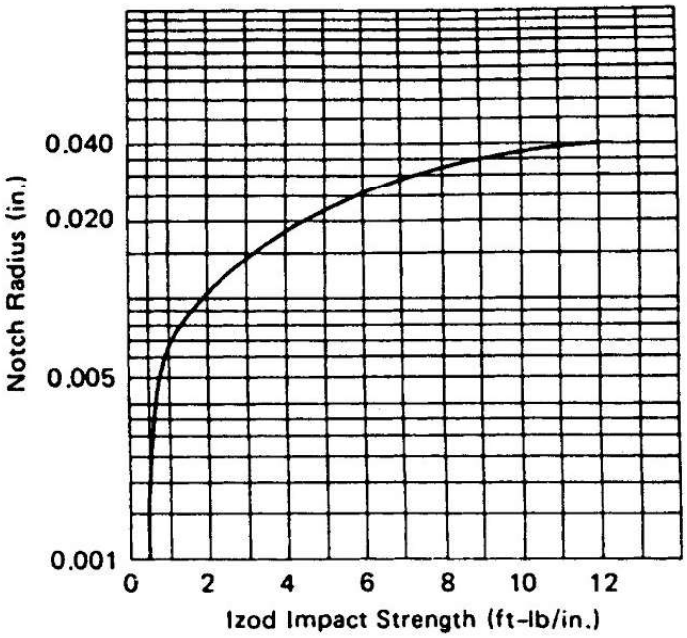


Fig. 8-6 Radius of notched Izod impact strength for nylon.

corners on the insides of parts are the most frequent property detractors.

The material data sheet should show the difference in impact strength between notched and unnotched test bars. In some materials this ratio is 1 to 30, but in others there is also a decided reduction in the strength of the notched bars. Some show no strength reduction, however. In a shaped product, an inside sharp corner is an indication that a certain specified tough material acts in a brittle manner. Sharp corners become stress concentrators.

The stress concentration factor increases as the ratio of the radius R to the part thickness T decreases. An R/T of 0.6 is favorable, and an increase in this value will be of some limited benefit. Properly counteracting certain other details in this problem will help in reducing stress concentration. In Fig. 8-5, we can see that a concentric radius, in addition to eliminating the outside sharp corner, can play an important part in holding down the value of the stress concentration.

The ASTM Izod impact strength of nylon with various notch radii is shown in Fig. 8-6. We see that with a radius of 0.005 in., the impact strength is about 1.3 ft-lb/in., with an R of 0.020 in., it is 4.5 ft-lb/in., and with an R of 0.040 in., it is 12 ft-lb/in. In most cases, a radius of 0.020 in. can be considered a sharp corner as far as end use is concerned, a size that is a decided improvement over 0- to 5-mil radius; therefore, it should be considered a minimum requirement and be so specified. If this radius of 0.020 in. causes interference, a corner such as that shown in Fig. 8-7 should be considered.

The recommended radius not only reduces the brittleness effect but also provides a streamlined flow path for the plastic in the mold. The radiused corner of the metal in the mold reduces the possibility of its breakdown and thus eliminates a potential repair need.

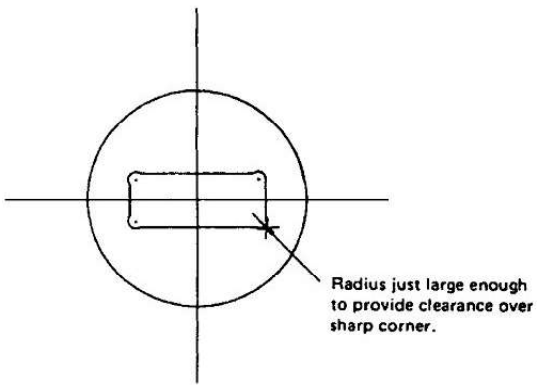


Fig. 8-7 Clearance radius for a sharp corner.

Spectrum Ex. 1014
IPR Petition - USP 11,536,632

Too large a radius is also undesirable because it wastes material, may cause sink marks, and may even contribute to stresses from having excessive variations in thickness.

Uniform Wall Thickness

Wall requirements are usually governed by the load, the support needs for other components, attachment bosses, and other protruding sections.

Designing a part to meet all these requirements while still producing a reasonably uniform wall will greatly benefit its durability. A uniform wall thickness will minimize stresses, differences in shrinkage, possible void formation, and sinks on the surface; it also usually contributes to material savings and economy in production.

Most of the features for which heavy sections are intended can be modified by means of ribbing, coring, and shaping of the cross section to provide equivalent strength, rigidity, and performance. Figure 8-8 shows a small gear manufactured from metal bar stock. The same gear converted to a molded plastic would be designed as shown in Fig. 8-9. This plastic gear design, compared to copying the metal gear, saves material; eliminates stresses from having thick and thin sections; provides uniform shrinkage in teeth and the remainder of the gear; avoids the danger of warpage, prevents bubble formation and potential weak spots with its thin web and tooth base; and having no sink in the middle of the thickness, provides full load-carrying capacity for the teeth.

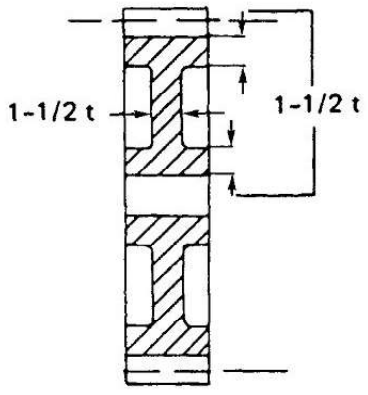


Fig. 8-8 Solid-steel gear.

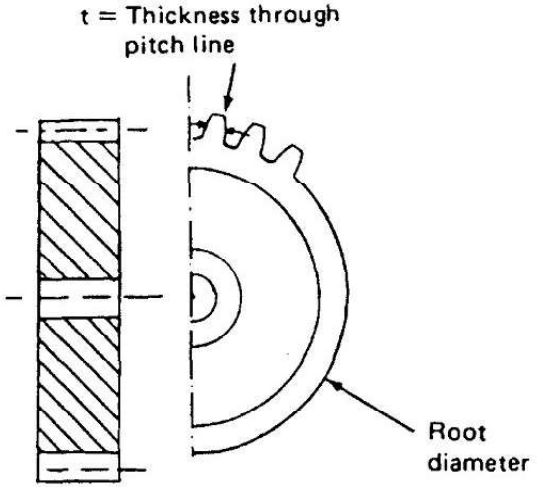


Fig. 8-9 Plastic design of the steel gear shown in Fig. 8-8.

Figure 8-10 illustrates both well- and poorly designed cross sections. If a case exists where some thickness variation is unavoidable, the transition should be gradual, to prevent sharp changes in temperature during solidification.

Wall Thickness Tolerance

When relatively deep parts are being designed, a tolerance for the wall thickness on the order of ±0.005 in. is usually given. This tolerance means that a product will be acceptable when made with this tolerance but that the wall thickness must be uniform throughout the circumference.

If we analyze the molding condition of such a part and assume that one side is made to minimum specifications and the opposite to maximum specifications, we find the following taking place: The resistance to plastic flow decreases to the third power of the thickness, which means that the thick side will be filled first, whereas the thin side will fill from all sides, instead of the gate side alone. This type of filling creates a pocket on the thin side and compresses the air and gases to such a point that the rising temperature resulting from compression causes the material to be charred while the pocket is filling up.

The charred material will create porosity, a weak area, and an electrically defective surface. Furthermore, the filling of the thick side

Spectrum Ex. 1014
IPR Petition - USP 11,536,632

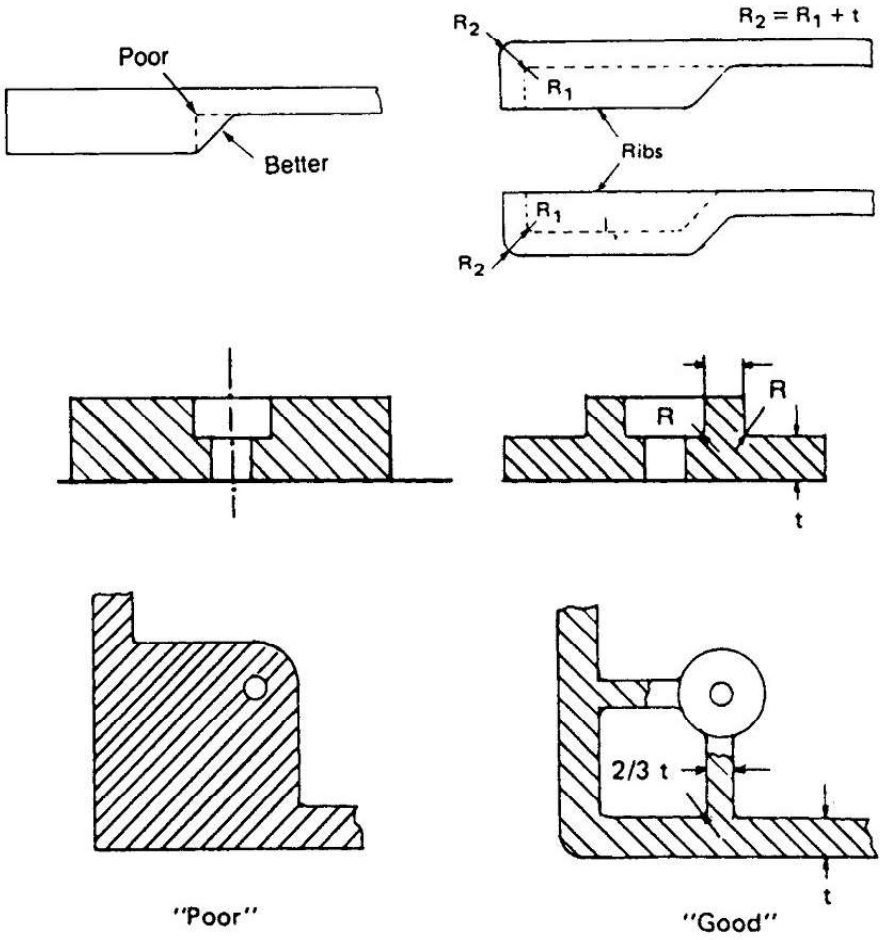


Fig. 8-10 Poor and better design features.

ahead of the thin side creates a pressure imbalance generated by the 5 to 10 tons/sq in. injection pressure that can cause the core to deflect toward the thin side, further aggravating the difference in wall thickness. This pressure imbalance will contribute to mold damage and make part production difficult if not impossible. We may conclude that the wall uniformity throughout the circumference must be within narrow limits, such as ±0.002 in., whereas the thickness, in general, may vary from the specified value by ±0.005 in.

Flow Pattern

Ultimately, part quality can be considered a direct outcome of a plastic melt's flow behavior in its mold cavity or cavities. Excessive restrictions and obstructions to the flow of material spell trouble in injection molding. Some examples of reduced-flow problems are illustrated in Figs. 8-11 through 8-19

Parting Lines

Parting lines (PLs) on the surface of a molded product, which are produced by the parting line of the mold, can often be concealed on a thin, inconspicuous edge of the part. Doing so preserves the good appearance of the molding and, in most cases, eliminates the need for any finishing. Figures 8-20 through 8-28 show parting line locations on various part configurations.

Gate Size and Location

Because of high melt pressure, the area near a gate is highly stressed, by both the

Spectrum Ex. 1014
IPR Petition - USP 11,536,632

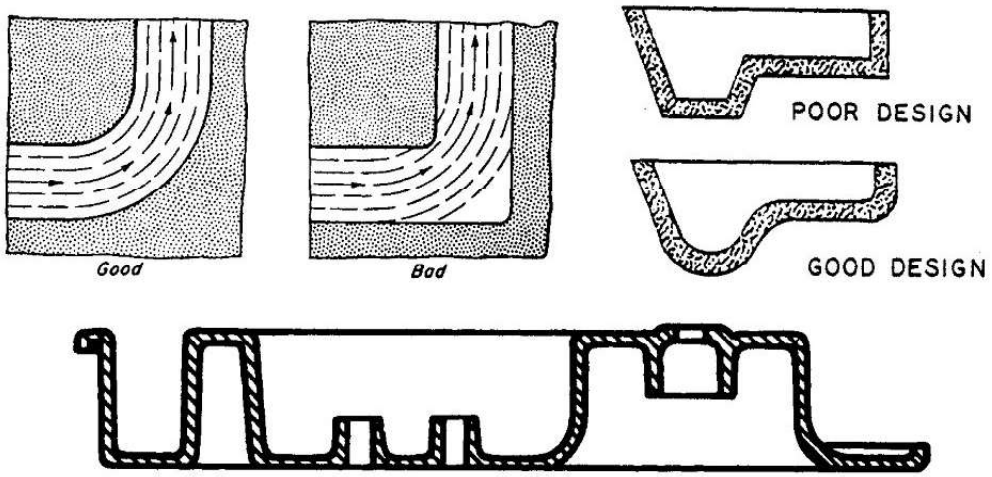


Fig. 8-11 Nominal thickness maintained throughout a part can simplify its melt flow. A good design has a minimum exterior radius of $1\frac{1}{2}$ times the wall thickness and a minimum of $\frac{1}{2}$ its thickness, which maintains uniform section thickness.

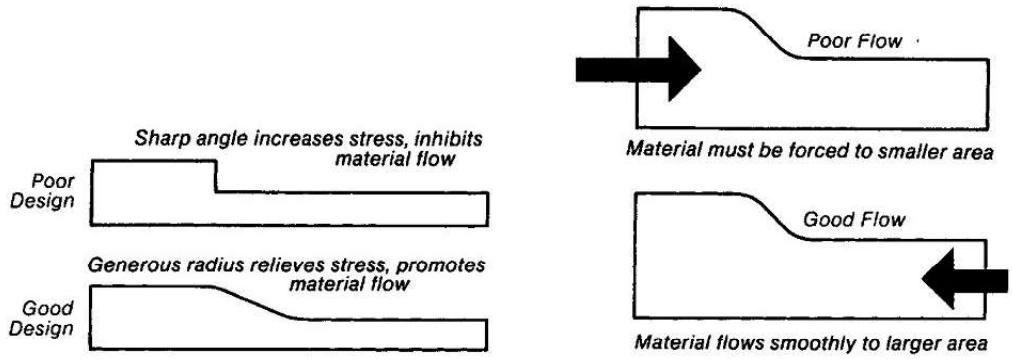


Fig. 8-12 Approaches to consider when making changes in wall thicknesses.

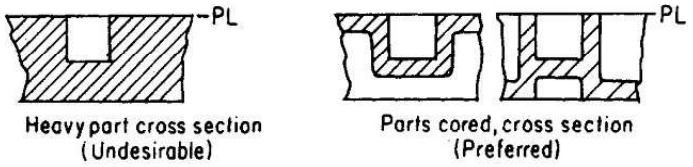


Fig. 8-13 Parts having heavy cross sections are subject to longer cycles and cures, laminations or skins, blisters, warpage, and increased fabricating costs.

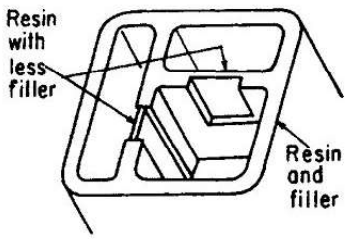


Fig. 8-14 When molding a reinforced plastic, thin sections sometimes lose strength, because fibers do not flow into a narrow space unless a suitable moldable material is used.

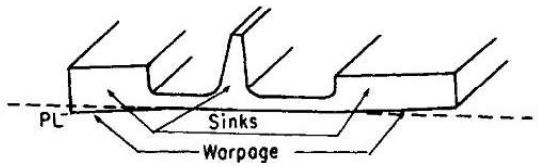


Fig. 8-15 Avoid having uneven sections, which cause distortion, warpage, cracks, sinks, and strains because of differences in shrinkage from one section to another. When this situation exists, the problem is normally eliminated by changing the process controls, which in turn usually requires a longer cycle time.

Spectrum Ex. 1014
IPR Petition - USP 11,536,632

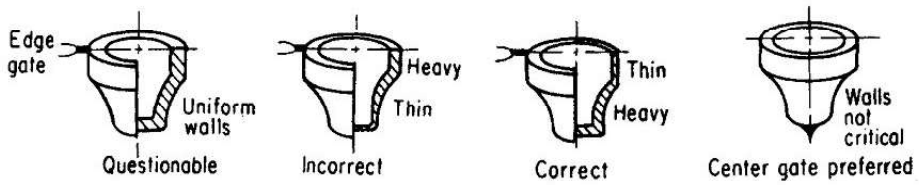


Fig. 8-16 Design of wall sections contributes to flow patterns and can be controlled to obtain the best end-product results.

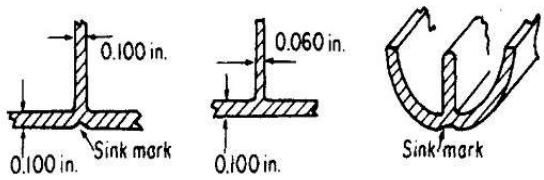


Fig. 8-17 Thicknesses of adjacent walls and ribs should be about 60% of the thickness of the main bodies to reduce the possibility of sink marks and promote better flow.

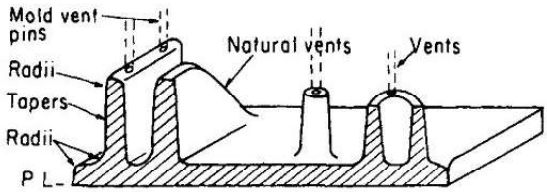


Fig. 8-18 Sharp corners, straight sides, and improper venting impede flow, resulting in strains and possible cracks. Optimum flow requires using maximum radii at corners, reasonable inside and outside tapers, as well as proper vents at mold parting lines, pockets, and blind holes.

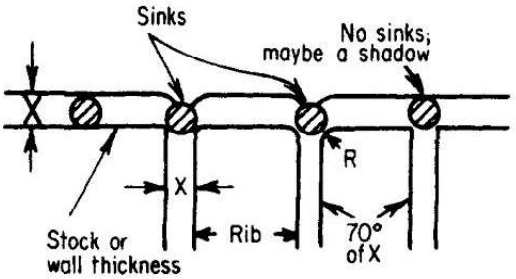


Fig. 8-19 Crude but effective method for locating sink marks and designing them "out" of plastics parts is, in effect, to "roll a ball" down the wall's thickness; if it permits a sink, the part will be marked.

frictional heat generated at the gate and high velocities of the flowing material. Using a small gate is desirable for separating the part from the feed line, but not for a part with low stresses. Gates are usually two thirds of a part's thickness. If they are that large or larger, they will reduce frictional heat, permit lower velocities, and allow the application of higher pressures for increasing the density of the material in the cavity. The product designer should caution the tool designer to keep the gate area away from the load-bearing surfaces and to make the gate size such that it will improve the quality of the product. Some examples regarding the design of gates are given in Figs. 8-29 through 8-34.

Taper or Draft Angle

It is desirable for any vertical wall of a molded product to have an amount of draft that will permit its easy removal from a mold. Figures 8-35 and 8-36 show two basic conditions in which draft is a consideration. The first example is the most desirable application of the draft angle. The amount of draft may vary from $\frac{1}{8}$ deg up to several degrees, depending on what the circumstances permit. A fair average may be from $\frac{1}{2}$ to 1 deg. When a small angle such as $\frac{1}{8}$ deg is used, the outside surface—the mold surface producing it—will require a high directional finish, to facilitate removal from the mold.

In the other example, as shown in Fig. 8-36, there is a separating inside wall that should generally be perpendicular to the base. The draft in this case would be on the low side ($\frac{1}{8}$ deg) so that additional material usage is small, the possibility of having voids close to the base is avoided, and increased cycle time in manufacturing is minimized. Here again,

Spectrum Ex. 1014
IPR Petition - USP 11,536,632

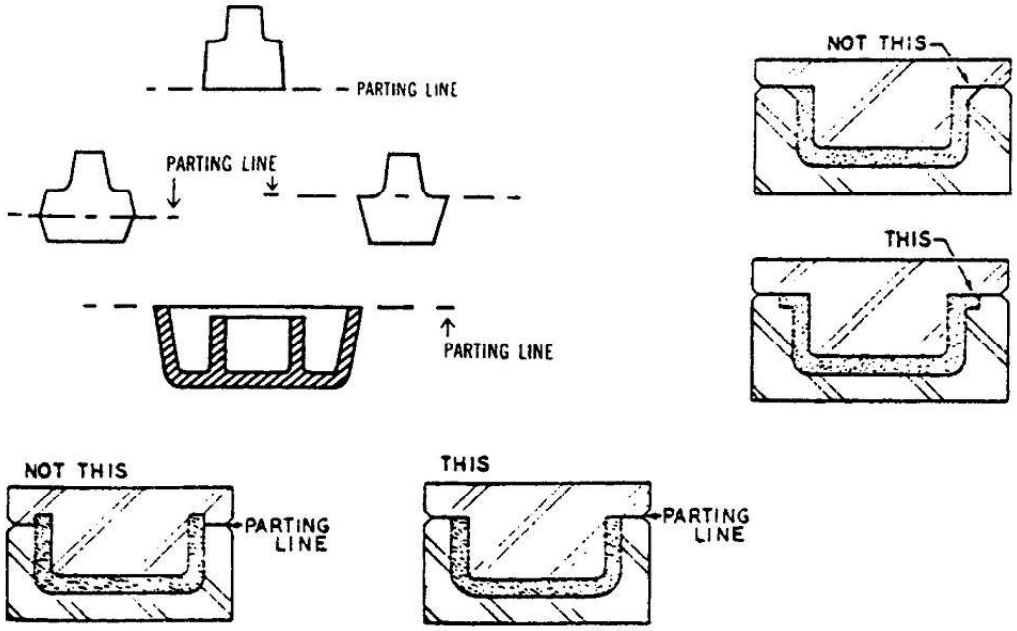


Fig. 8-20 Examples of parting line locations. The extreme right diagram shows that having sharp corners could cause poor dimensional control or possibly nonflat surfaces with inadequate mold cooling.

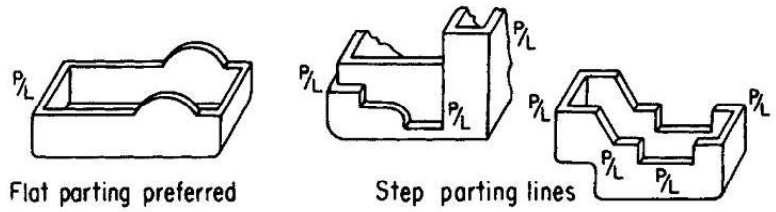


Fig. 8-21 Consider the parting lines when designing parts that may require odd or different radii, contours, and stepped parting lines.

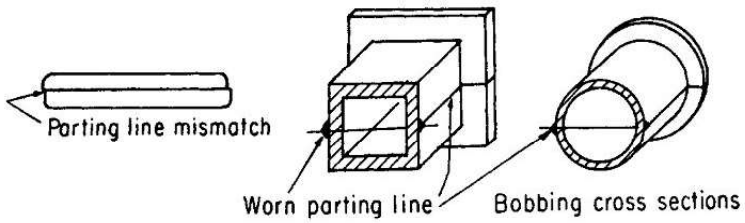


Fig. 8-22 Excessive wear on mold parting lines can create a mismatch on a molded part that will appear greatly exaggerated [a 0.0003-in. (0.0008-cm) mismatch can appear to be 0.020 in. (0.05 cm)] or can create burrs from 0.001 to as much as 0.010 in. (0.003 to 0.025 cm).

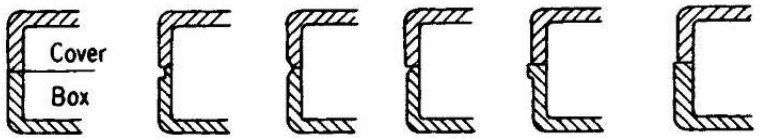


Fig. 8-23 Example of a series of possibilities for designed mismatches.

Spectrum Ex. 1014
IPR Petition - USP 11,536,632

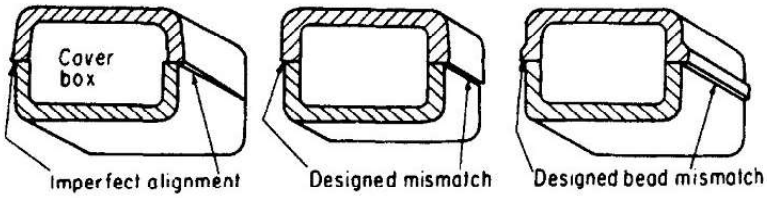


Fig. 8-24 Mismatch in alignment of two molded parts (such as a box and cover) usually is traceable to part warpage or difference in shrinkage; the misalignment at the parting line appears to be improved when a bead or designed mismatch is utilized.

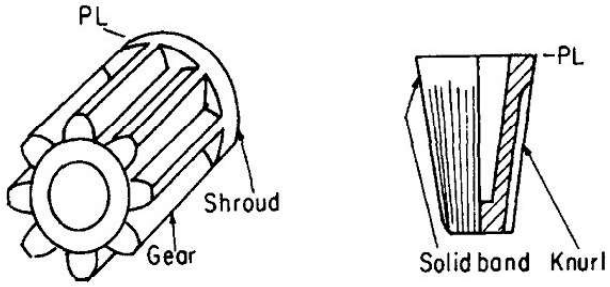


Fig. 8-25 Allowing a parting line flash in gear teeth and on knurled parts requires expensive deflashing operations. To prevent flash, add a shroud to the gear or solid band or bead to the knurl at the mold parting line.

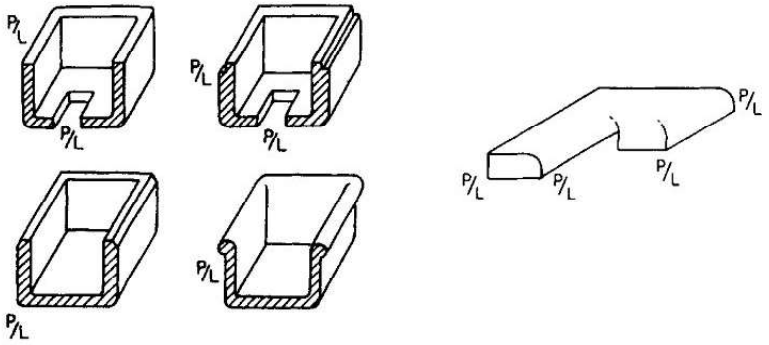


Fig. 8-26 Four possibilities for locating parting lines on assembly parts, boxes, and covers while maintaining good aesthetics.

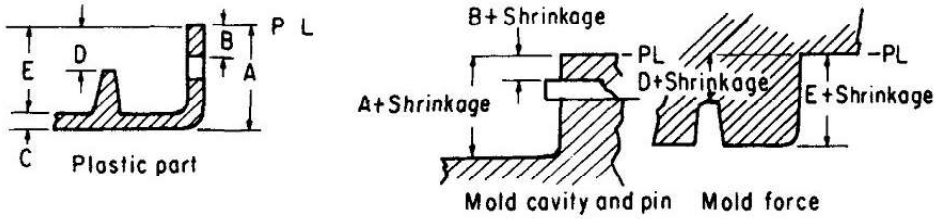


Fig. 8-27 Dimensions and tolerances given for parting lines should include allowances for potential flashing conditions. All molds for TPs or TSs can open and flash to some extent, depending on the mold design and construction, melt pressure, mold-clamping force, and type of material. For example, if the flash is 0.003 in. (0.008 cm) at the mold parting lines, part dimensions A, B, C, D, and E will increase 0.003 in. (left diagram). The drawing at the right shows how to estimate mold components.

Spectrum Ex. 1014
IPR Petition - USP 11,536,632

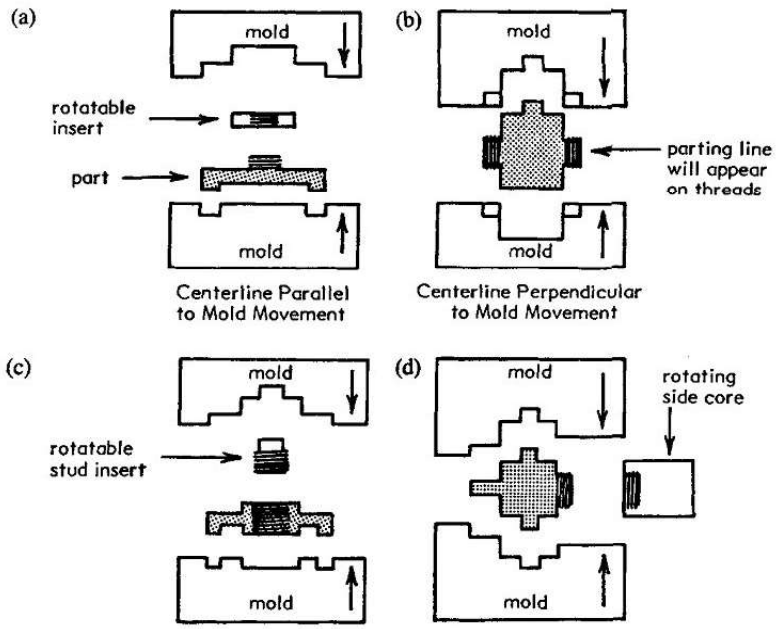


Fig. 8-28 Parting lines and molded threads in various positions.

the vertical molding surfaces will demand a much higher surface finish, with polishing lines in the direction of part withdrawal. On shallow walls, the draft angle can be considerably larger, since the influence of the enumerated drawbacks will be minor. The designer should be cognizant of the need for drafts in vertical walls. If problems are encountered during the removal of parts, stresses can result, the shape of the product can be distorted, and surface imperfections introduced.

Some examples of different approaches to the design of drafts are shown in Figs. 8-37 through 8-40. One of the difficulties when applying draft to a part is the creation of heavy walls. The potential problem of removal can be remedied by using parallel drafts on walls, as shown in Fig. 8-40, where the walls are kept uniform. A guide to determining the amount of dimensional change required due to the draft angle is given in Fig. 8-41.

Weld Lines

With molded parts that include openings (holes), problems develop. In the process of filling a cavity, the flowing plastic is obstructed by the core, splits its stream, and surrounds the core. The split stream then reunites and continues flowing until the cavity is filled. The rejoining of the split streams forms a weld line that lacks the strength properties that exist in an area without a weld line because the flowing material tends to bring air, moisture, and lubricant into the area where the joining of the stream takes place and

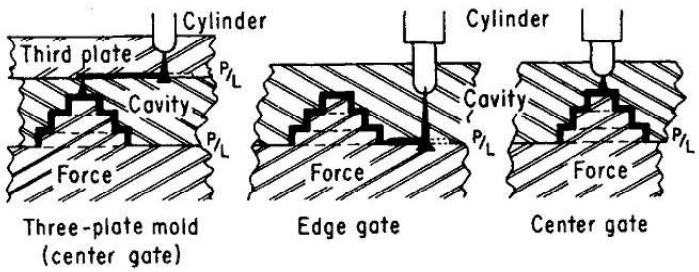


Fig. 8-29 Injection molds are especially suitable for producing parts to meet different mold styles, as shown in these three examples.

Spectrum Ex. 1014
IPR Petition - USP 11,536,632

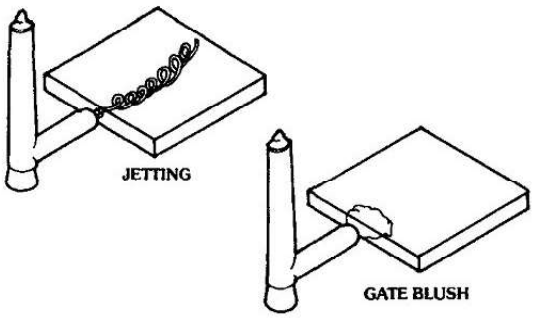


Fig. 8-30 Examples of some melt flows using too small a gate. The melt should fill the cavity uniformly.

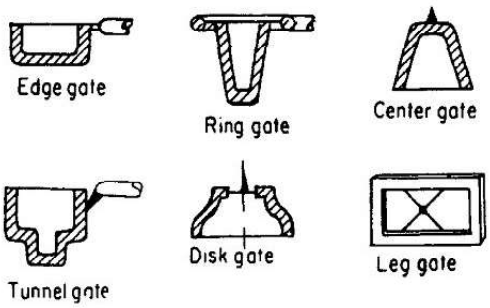


Fig. 8-31 Among the various methods available for gating are edge, ring, center, tunnel, disk, and leg gates. Consider the material and design factors before selecting the type of gate.

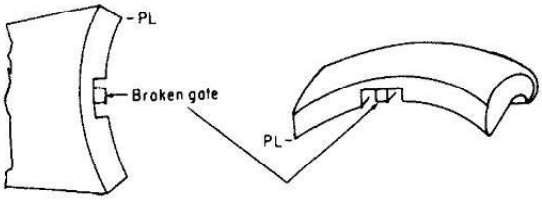


Fig. 8-32 Gate breaking and clipping are low-cost removal methods when the part design permits locating a recessed gate in a hidden area, as in this handle. This type of problem is eliminated by using hot-runner molds for TPs or cold-runner molds for TSs.

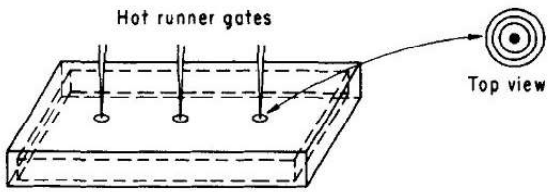


Fig. 8-33 In molding a large, flat TP surface using hot or insulated runners, having one or more small gates can help reduce warpages. One way to camouflage the resulting gate scars is to use the bull's-eye design shown (with hot runners no gate problem develops).

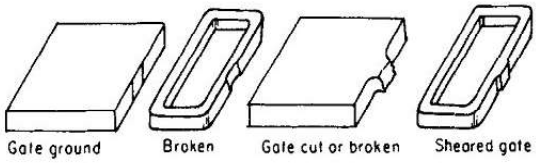


Fig. 8-34 Some gates must simply be cut, sheared with fixtures, ground, or finished, but those broken off from the parts to save finishing costs could retain their recesses or protrusions.

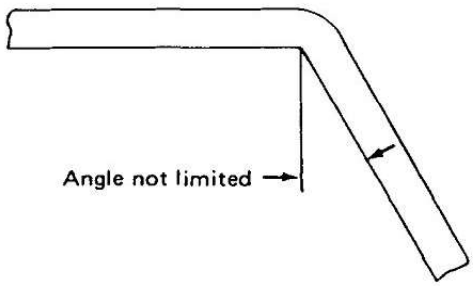


Fig. 8-35 External wall taper.

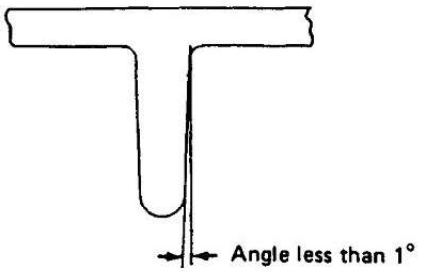


Fig. 8-36 Internal wall taper.

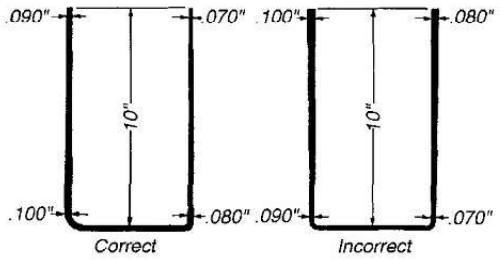


Fig. 8-37 Example of side-wall taper for long-draw products.

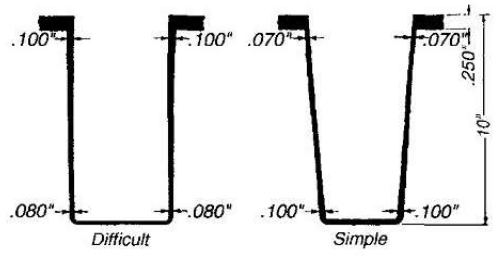


Fig. 8-38 Example of designs for long-draw products. The left view has little clearance, but the right side is simplified by the clearance allowed by the walls' taper.

Spectrum Ex. 1014
IPR Petition - USP 11,536,632

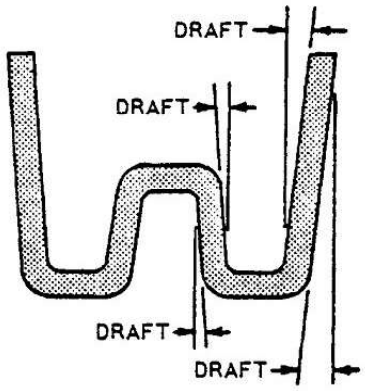


Fig. 8-39 Example of using multiple drafts to permit ease of part removal.

introduces foreign substances into the welding surface. Furthermore, since the plastic material has lost some of its heat, the temperature for self-welding is not conducive to the most favorable results. A surface that is to be subjected to load-bearing should not contain weld lines. If this is not possible, the allowable working stress should be reduced by at least 15%. Some examples of different aspects pertaining to weld lines are shown in Figs. 8-42 and 8-43.

Meld Lines

A meld line is similar to a weld line except that the flow fronts move in parallel rather than meet head on (Fig. 8-44).

Vent, Trapped Air, and Ejector

Vents can be used to release air entrapped in cavities as well as moisture and gases formed from melts that were not previously removed during processing (Chap. 4). Vents and ejectors may be considered when designing such parts, as shown in Figs. 8-45 through 8-51.

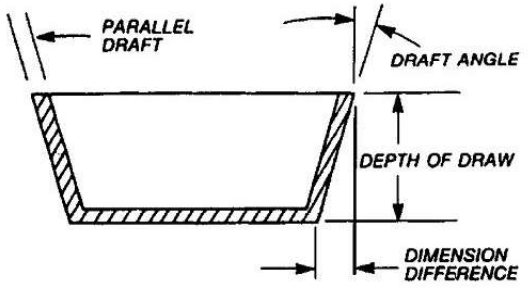


Fig. 8-40 Using parallel drafts with heavy walls.

Undercuts

Undercuts, whether external or internal, should be avoided if possible. In cases where it is essential to incorporate them in part design, a great many can be fashioned by appropriate mold design in which either sliding components on tapered surfaces or split cavity cam actions will produce the needed undercut. This obviously means increased tool cost, in the neighborhood of some 15 to 30%.

Some conditions will, however, permit incorporating undercuts with conventional stripping of the part from the mold. Certain precautions are necessary to attain satisfactory results. First, the protruding depth of the undercut should be two-thirds of the wall thickness or less. Second, the edge of the mold against which the part is ejected should be radiused to prevent shearing action. Finally, the part being removed should be hot enough to permit easy stretching and return to its original shape after removal from the mold (Figs. 8-52 through 8-61).

How easily the task can be accomplished depends on the material's elasticity and springback. Many threaded plastic caps are stripped from the cores instead of being unscrewed. Coarse threads with the crest of the core thread rounded and a material with good elongation and the ability to spring back make it feasible to apply conventional part stripping. The undercut problem can be solved by the cooperation of the designer, moldmaker, and processor, since each product configuration presents different possibilities.

Blind Holes

When molding products that include holes, it is important to ensure that sufficient

Spectrum Ex. 1014
IPR Petition - USP 11,536,632

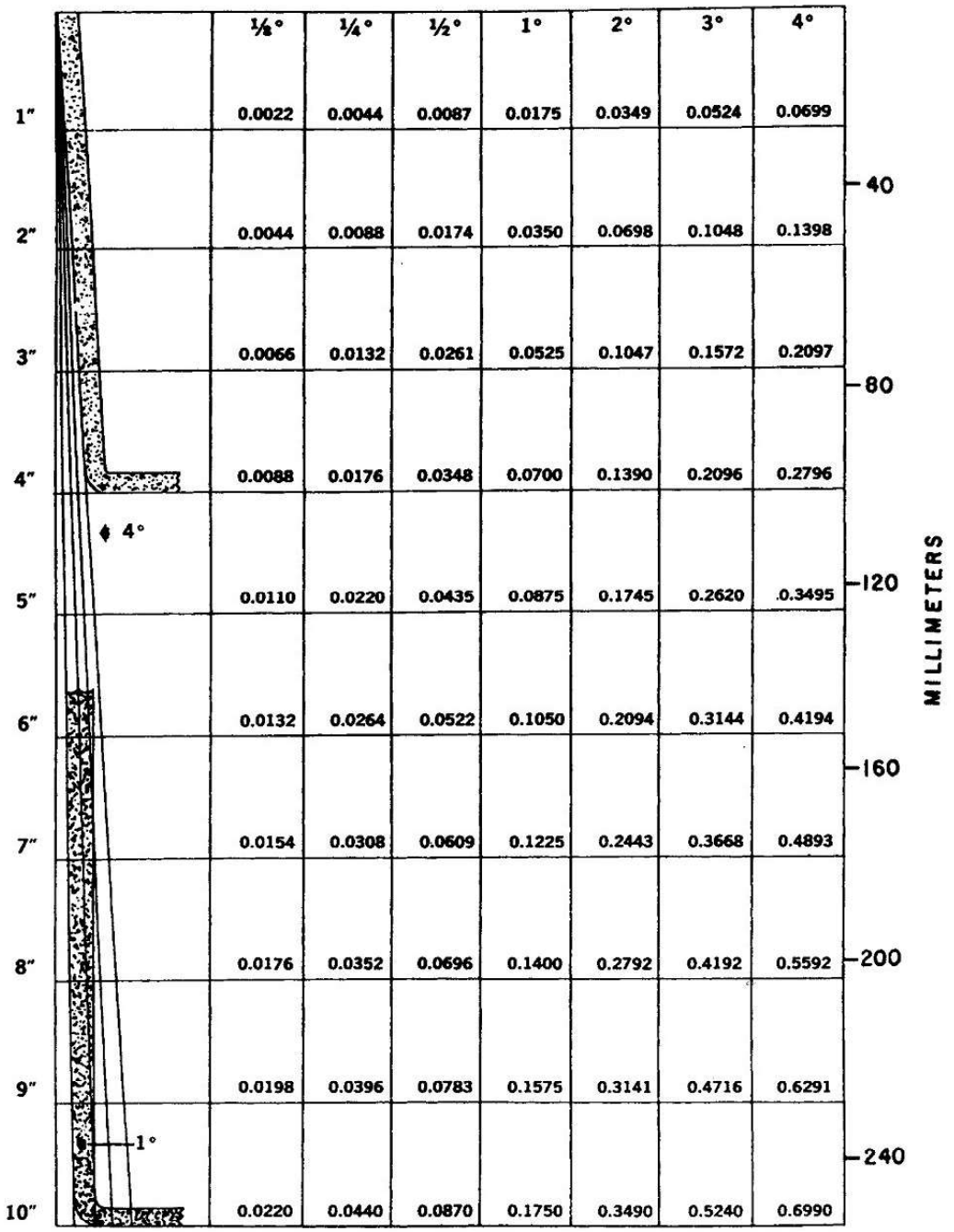


Fig. 8-41 Relation of the degree of taper per side to the dimension in in./in.

material surrounds the holes and that the melt flows properly. Some examples of approaches and problems that can develop are shown in Figs. 8-62 through 8-69.

A core pin forming blind holes is subjected to the bending forces that exist in the cavity, owing to the high melt pressures. Calculations can be made for each case by establishing the core pin diameter, its length, and the anticipated pressure conditions in the cavity. From technical handbooks, we know that a pin supported on one end only will deflect 48 times as much as one supported on both ends. This suggests that to obtain a

Spectrum Ex. 1014
IPR Petition - USP 11,536,632

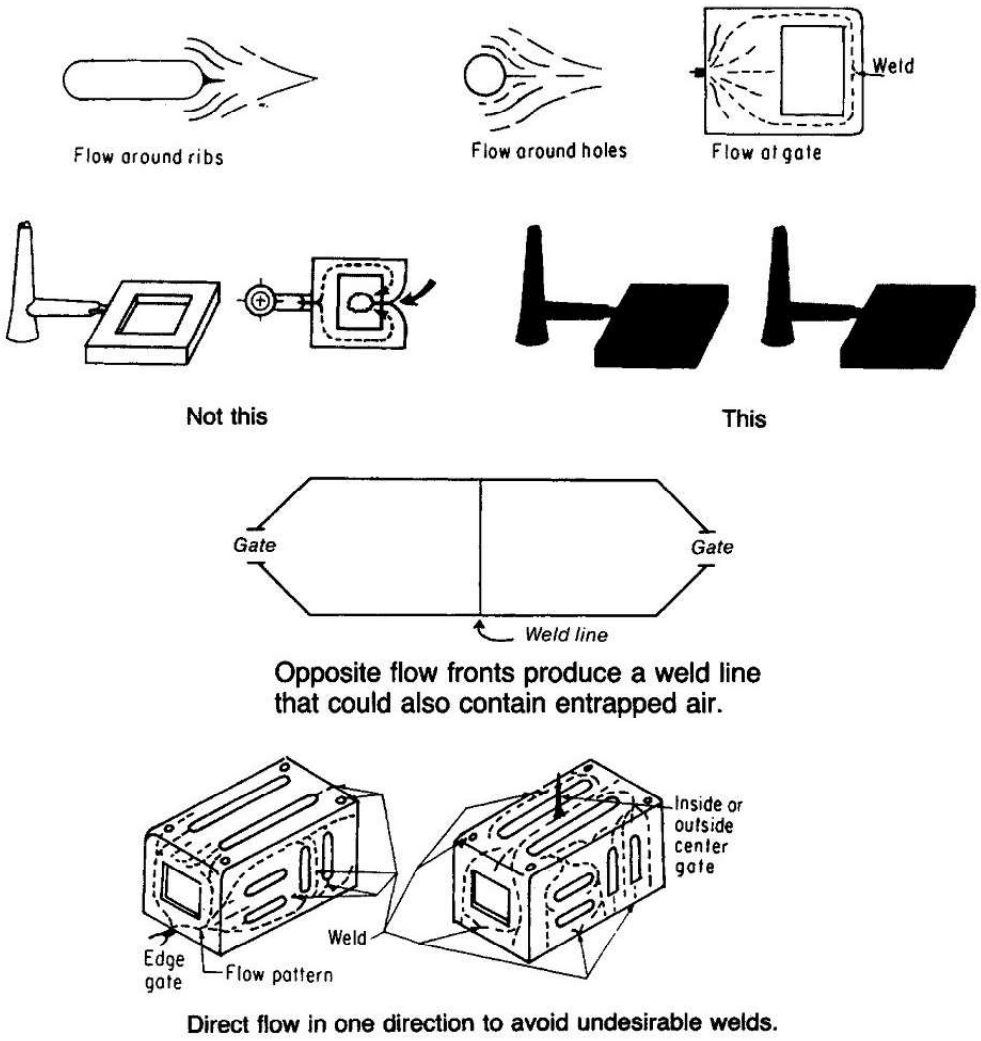


Fig. 8-42 Some examples of flow patterns to consider during the design state to eliminate or at least minimize weld lines to obtain maximum strength.

straight hole the depth of hole in relation to diameter should be small. Sometimes a deep, small-diameter hole is needed, as in pen and pencil bodies. In this case, the plastic flow is arranged to hit the free end of the core from four to six evenly spaced gates. This will cause a centering action, and the plastic will continue flowing over the diameter in an umbrellalike pattern to balance the pressure forces on the core. When this type of flow

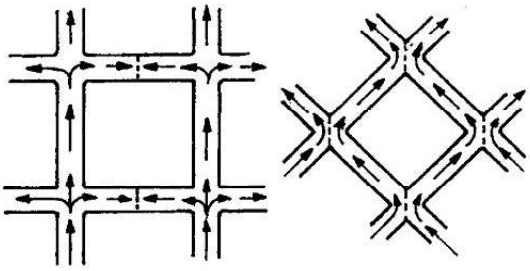


Fig. 8-43 Different flow patterns develop different weld lines.

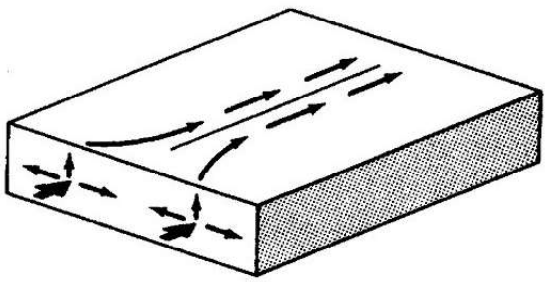


Fig. 8-44 Meld line.

Spectrum Ex. 1014
IPR Petition - USP 11,536,632

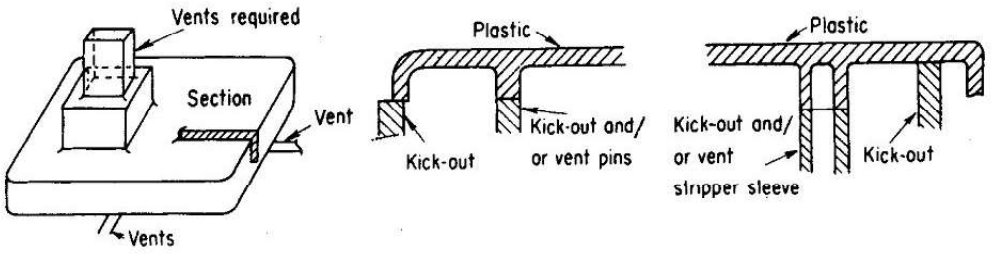


Fig. 8-45 Parting lines can provide venting, but in deep pockets kick-out pins or vent pins incorporated into a mold can vent or remove air or gas.

pattern is impractical, an alternative may be a through hole or tube formation combined with a postmolding sealing or closing operation by spinning or ultrasonic welding.

At the other extreme, consider a $\frac{1}{4}$-in.-diameter core exposed to a pressure of 4,000 psi with an allowance for deflection of 0.0001 in. and see how deep a blind hole can be molded under these conditions.

According to engineering handbooks, the deflection a may be used to calculate this depth l as follows:

$$a = \frac{Wl^3}{8EI}$$

$$= \frac{1{,}000l^4}{8 \times 3 \times 10^7 \times 0.049 \times 0.0039}$$

$$I = \frac{\pi d^4}{64} = 0.049d^4$$

$$d^4 = 0.0625 \times 0.0625 = 0.0039$$

where W = total band
= psi $\times$ d $\times$ l (projected area of pin)
$= 4{,}000 \times \frac{1}{4} \times l = 1{,}000l$
l = length of pin
d = diameter of pin

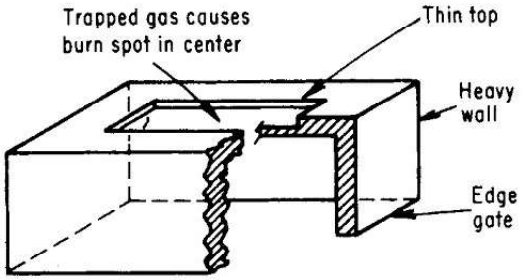


Fig. 8-47 Here the melt flow passes through heavy sections first, pushing air and gases into the topmost thin sections, where they become trapped. This design should either be center-gated at its top or use thinner walls, if possible.

a = deflection = 1/10,000

$$\frac{1}{10{,}000} = a$$

$$= \frac{1{,}000l^4}{(8 \times 3 \times 10^7 \times 0.049 \times 0.0039)}$$

= deflection

$$l^4 = 0.0045864$$

$$l = 0.26 \text{ in.}$$

or slightly over diameter size

If a hole deeper than 0.26 in. is needed, we can calculate the amount of deflection that will be present and whether the calculated deflection will produce an opening of the

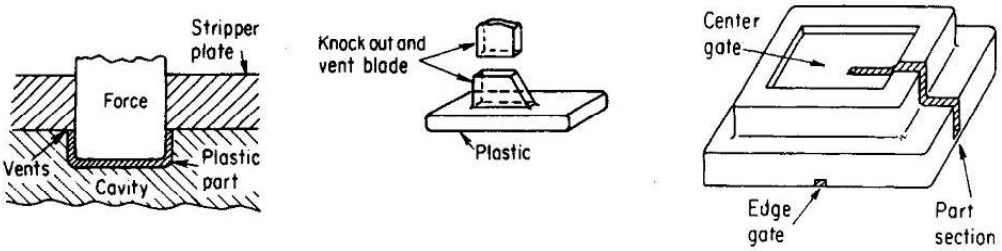


Fig. 8-46 Center-gate this type of part, if possible. If aesthetics prevent this approach and edge gating is requested, the material will flow around the edge and to the thin top section last. Thus, air—and any gas that may be present because of melt—will be trapped in the top, which may not be acceptable.

Spectrum Ex. 1014
IPR Petition - USP 11,536,632

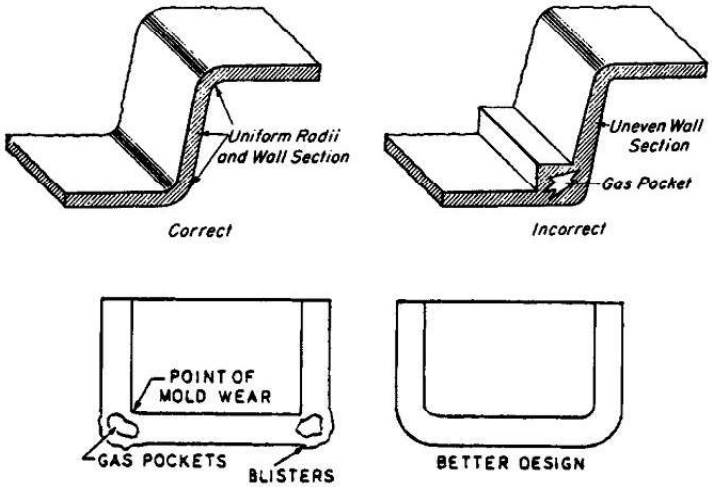


Fig. 8-48 These cross-sectional views show the results of uniform and nonuniform wall thicknesses in eliminating air and gas entrapment.

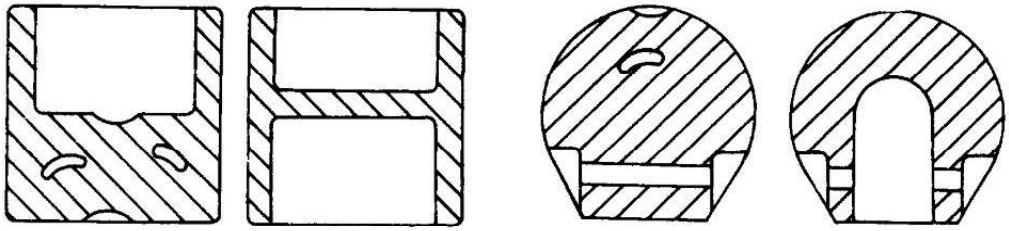


Fig. 8-49 To aid in eliminating air and gas entrapment, aim to reduce heavy or thick sections like those on the left of each pair.

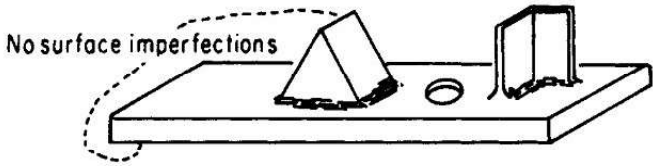


Fig. 8-50 Gates and ejector pins create marks and blemishes. To aid in properly locating gates and ejector pins, parts drawings should indicate surfaces that can be marred without creating problems.

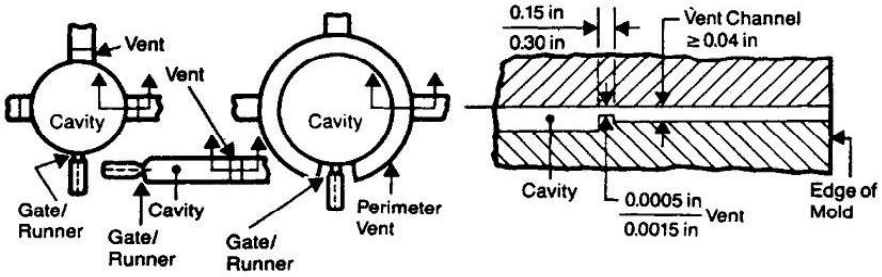


Fig. 8-51 Schematics of mold vent designs.

Spectrum Ex. 1014
IPR Petition - USP 11,536,632

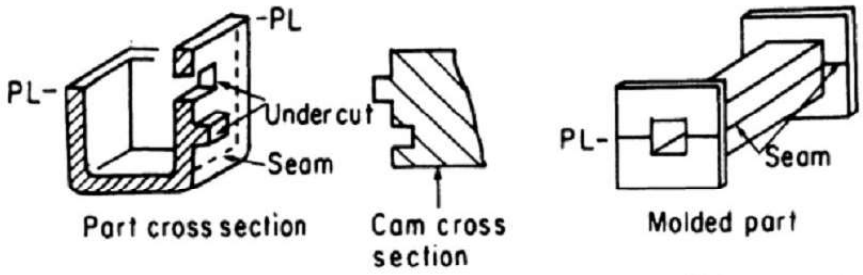


Fig. 8-52 Split mold with mold sliding-cam actions for outside undercuts will show seams in the molded part that may be objectionable. These can be partially hidden by locating them in the least conspicuous areas when designing the molds.

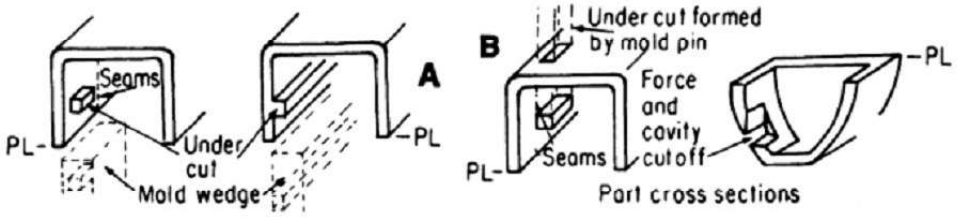


Fig. 8-53 Inside undercuts can be made with removable wedges or ejector pins and ejector wedges, but seams will still show. The left view shows undercuts molded with removable wedges; the right view shows the mold knock-out pin positioned through the permissible opening in the part and cavity cutoff approaches.

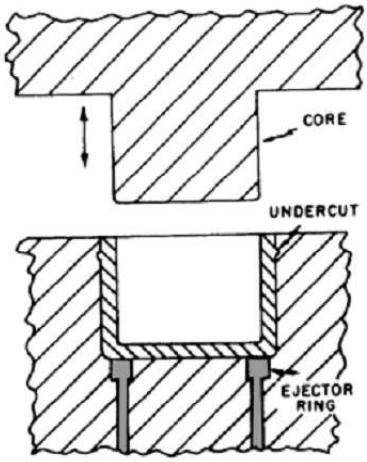


Fig. 8-54 Mold for external undercuts to retain part in the required female cavity prior to ejection. When the part must be retained on a male cavity, undercuts are on the male plug.

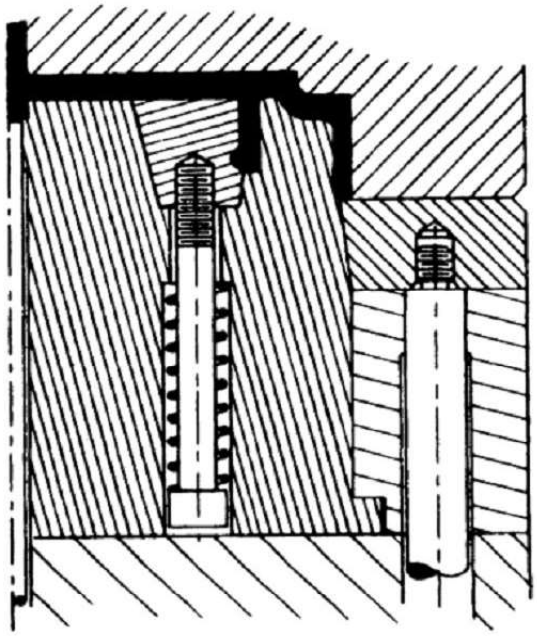


Fig. 8-55 Example of an undercut made possible by using an ejector pin.

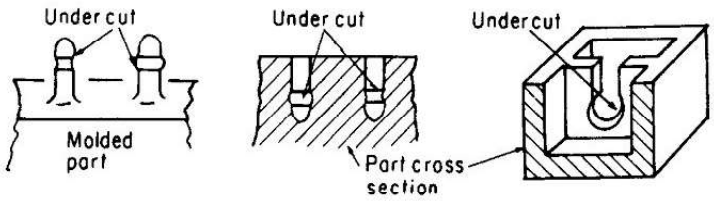


Fig. 8-56 The goal should be to avoid undercuts wherever possible, especially in small holes and on projections. With rigid plastics, undercuts usually require complex, more expensive molds and molding operations. However, with flexible types of plastics, stripping the mold will permit ease of removal from the mold. This action can be used with rigid plastics when it is practical to remove a part while the plastic is still soft, just prior to removing the entire part.

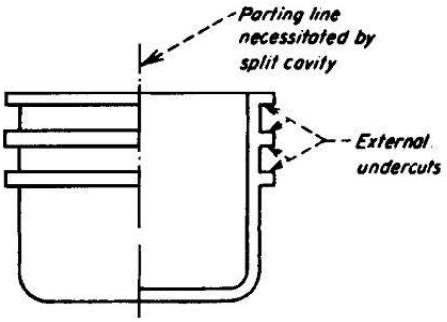


Fig. 8-57 External undercuts with this split-mold design eliminate the undercut problem but the part develops a parting line.

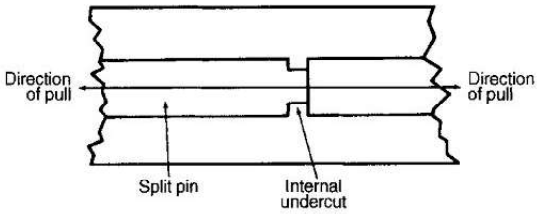


Fig. 8-59 Split-pin molding for internal undercuts.

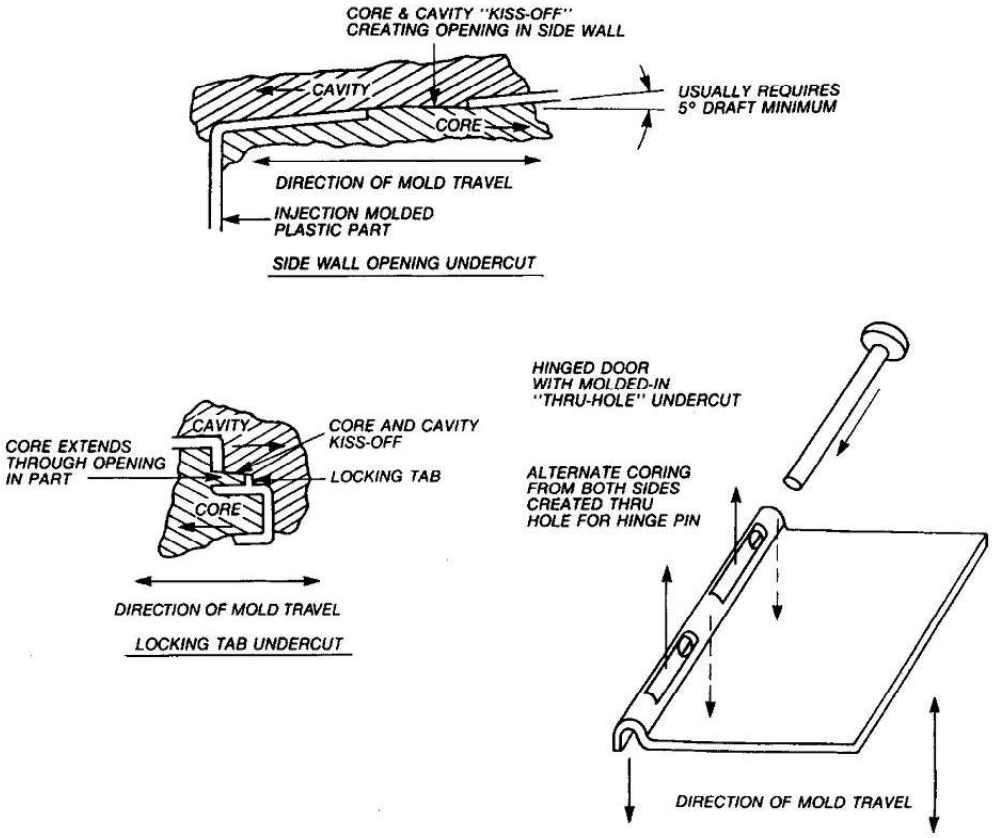


Fig. 8-58 Creating undercuts with simple tooling.

Spectrum Ex. 1014
IPR Petition - USP 11,536,632

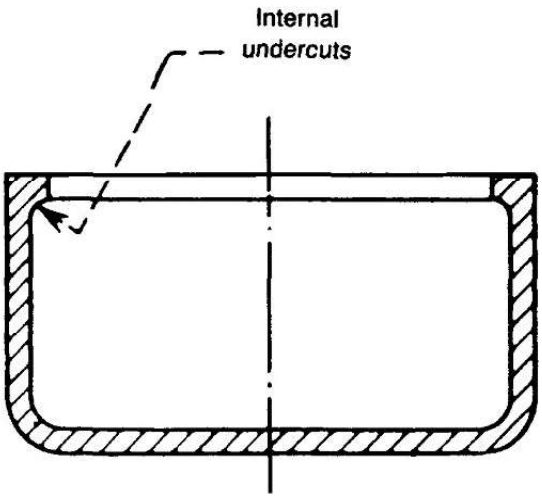


Fig. 8-60 Internal undercuts, as shown, can be provided and it may be possible to snap or strip the part from the mold. Otherwise, a core pulling device is required.

necessary tolerance, as well as the kind of stress that will be generated in the pin, along with its corresponding life expectancy. Let us now assume that the desired depth of hole is $\frac{3}{8}$ in.

The deflection is then calculated as follows:

$$a = \frac{Wl^3}{8EI}$$

$$= \frac{1{,}000l^4}{8 \times 3 \times 10^7 \times 0.049 \times 0.0039}$$

$$l^4 = 0.0198$$

$$a = \frac{0.0198}{45{,}864}$$

$$= 0.00043 \text{ in. deflection}$$

The maximum stress S is found by

$$S = \frac{Wl}{2Z}$$

$$= \frac{1{,}000l^2}{2 \times 0.006125}$$

$$= \frac{1{,}000 \times 0.1406}{2 \times 0.006125}$$

$$Z = 0.098d^2 = 0.006125$$

$$S = 11{,}480 \text{ psi}$$

These results indicate that a hole with a 0.0004-in. variation may be satisfactory, and if the pin is made of a springlike material and properly heat-treated, it should last a long time.

Bosses

Bosses and other projections from the nominal wall are commonly found in injection-molded plastic parts. These often serve as mounting or fastening points. Figure 8-70 shows some typical boss designs, along with common problems. As with rib design, avoiding overly thick wall sections is important, to minimize the chance of appearance or molding problems. When bosses are designed to accommodate self-trapping screws, the inside diameter and wall thickness must be controlled to avoid excessive buildup of hoop stresses in the boss. Ribs are frequently used in conjunction with bosses when lateral forces are expected. Special care must be used with tapered pipe threads, since they can create

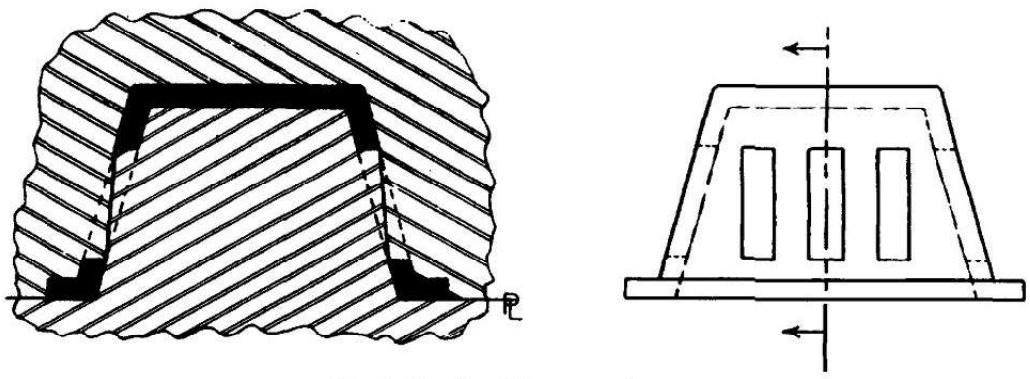


Fig. 8-61 Avoiding an undercut.

Spectrum Ex. 1014
IPR Petition - USP 11,536,632

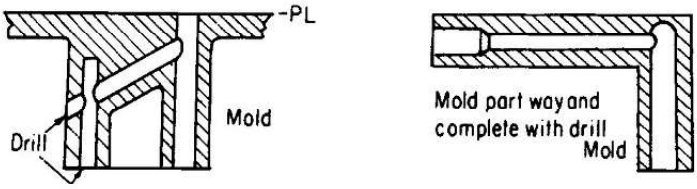


Fig. 8-62 Holes impractical to mold must be drilled, but they must not be so close to edges or corners that cracks result. A small-diameter hole is difficult to drill along its intended direction to any great depth, so the most practical approach in many products is to mold it part way and then drill it the remainder of the distance.

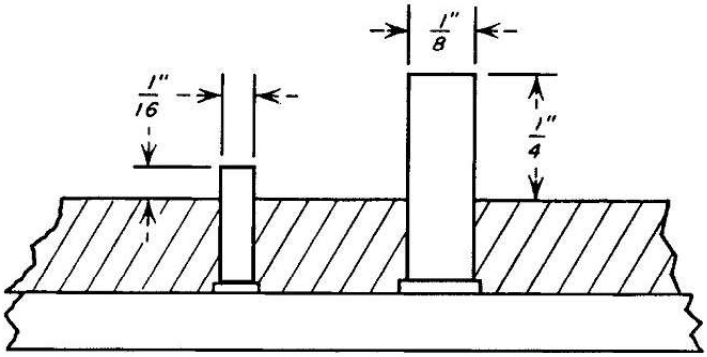


Fig. 8-63 Basic guide for good blind-hole design.

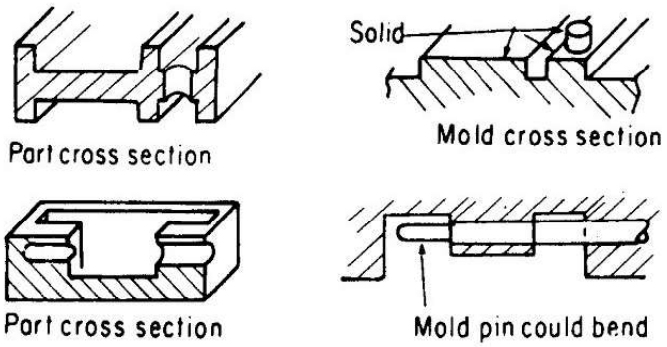


Fig. 8-64 When holes are too near an edge or corner, material may not "weld" properly around mold pins. Also, the flow of the melt can bend mold pins for blind holes when their length exceeds a diameter of $2\frac{1}{2}$ times and when holes are to be long with small diameters, even if these are anchored at both ends.

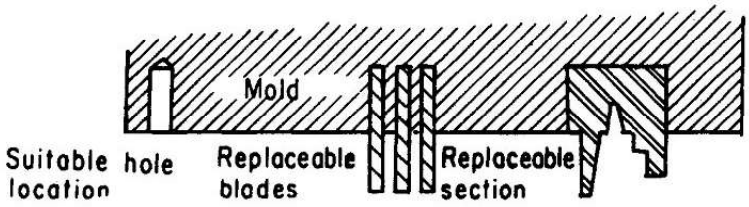


Fig. 8-65 When delicate parts are molded, mold designs should provide for proper hole locations and replaceable blades and members in wall sections.

Spectrum Ex. 1014
IPR Petition - USP 11,536,632

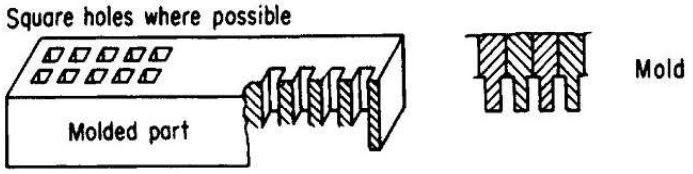


Fig. 8-66 Whenever possible with close-tolerance parts, consider the potential of using a laminated type of mold.

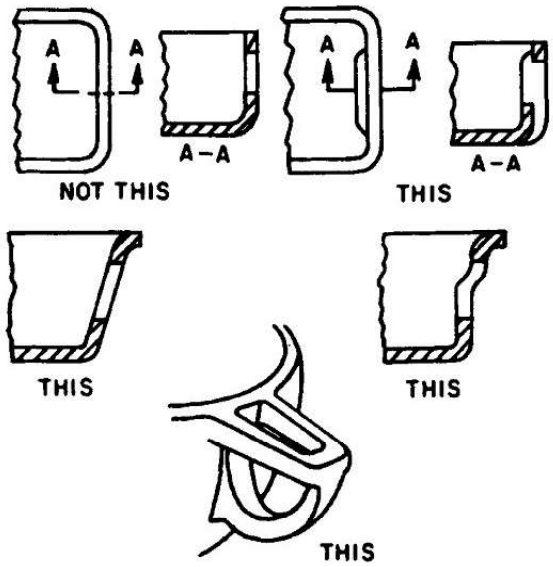


Fig. 8-67 Methods for molding holes or openings in side walls without undercutting mold movement.



Fig. 8-68 Whenever possible, chamfering should be used on open holes, because it reduces or eliminates the potential for rough molded corners, cracks, and the like.

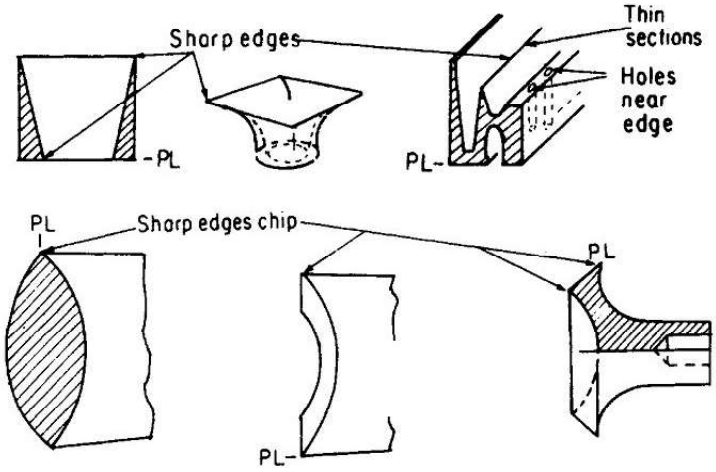


Fig. 8-69 Avoid part cross sections that are so thin that they will be prone to cracking, sharp edges that will chip or break, and holes near enough the edges to cause them to chip. Sections should also not be so thin that melt will flow and weld in their thin edges. Certain plastics are prone to this type of action.

Spectrum Ex. 1014
IPR Petition - USP 11,536,632

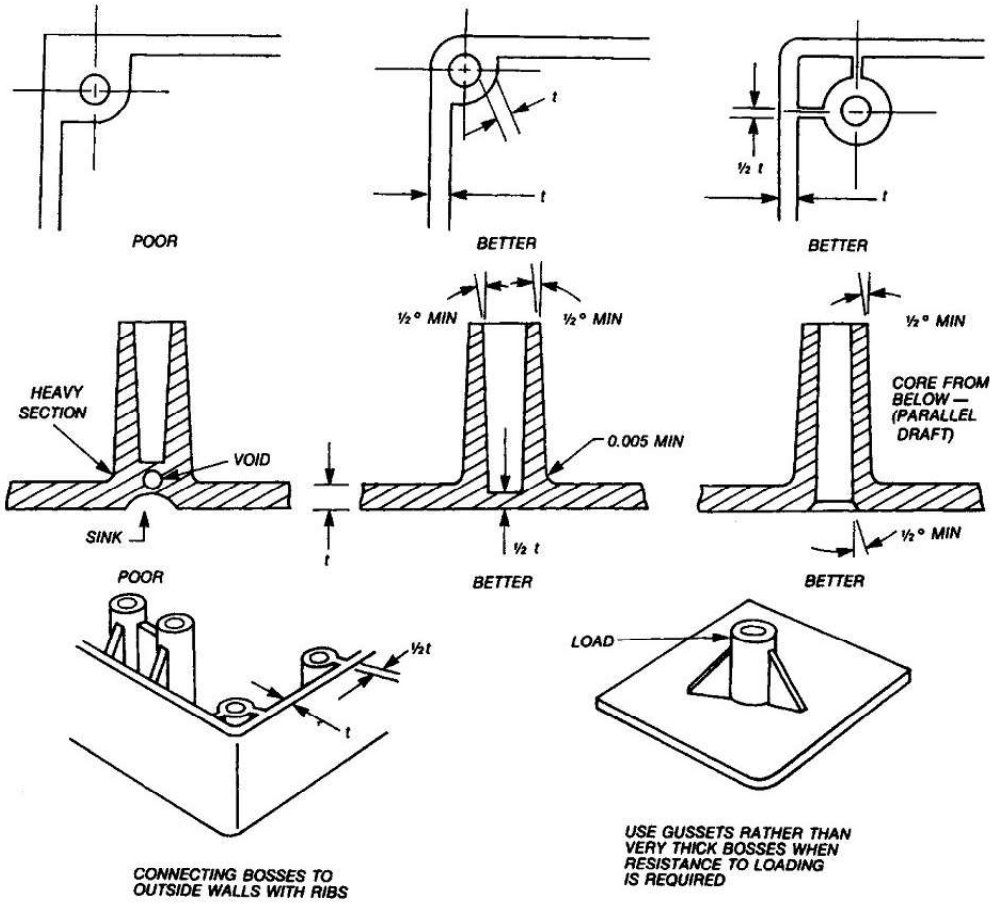


Fig. 8-70 Design guides for molding bosses.

a wedging action on the boss. If there is a choice, the male rather than the female pipe thread should be the one molded into the plastic.

Coring

The term *coring* in injection molding refers to the addition of steel to the mold for the purpose of eliminating plastic material in that area. Usually, coring is necessary to create a pocket or opening in the part, or simply for the purpose of reducing an overly heavy wall section (Fig. 8-71). For simplicity and economy in injection molds, cores should be parallel to the line of draw in the mold. Cores placed in any other direction usually create the need for some type of side action (either a cam or hydraulic cylinder) or manually loaded and unloaded loose cores.

Blind holes in molded plastic parts are created by a core supported by only one side of the mold. The length of the core and depth of the hole are limited by the ability of the core to withstand the bending forces produced by the flowing plastic without excessive deflection. For this reason, the depth of a blind hole should not exceed three times its diameter or minimum cross-sectional dimension. For small blind holes with a minimum dimension below $\frac{1}{4}$ in., the L/D ratio should be kept to 2. With through holes the cores can be longer, since they are supported by the opposite side of the mold cavity. Sometimes, the cores can be split between the two sides and interlocked when the mold is closed, allowing for the creation of long through holes. With through holes, the overall length of a given-size core can generally be twice as long as that of a blind hole. In some instances, even longer cores are necessary. The tool can be designed

Spectrum Ex. 1014
IPR Petition - USP 11,536,632

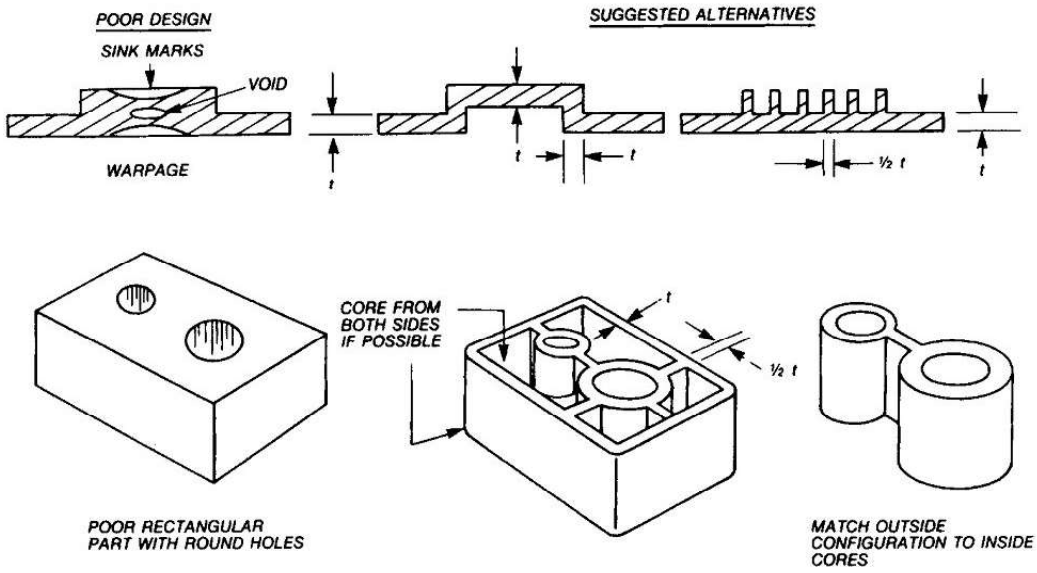


Fig. 8-71 Examples of coring in injection molds.

to balance the hydraulic pressure on the core pin, thus limiting the deflection.

Press Fits

Press fits, which depend on having a mechanical interface, provide a fast, clean, economical assembly. A common usage is to have a plastic hub or boss that accepts either a plastic or metal shaft or pin. The press-fit procedure tends to expand the hub, creating tensile or hoop stress. If the interference is too great, high strain and stress will develop. The plastic part will do one of several things. It may fail immediately, by developing a crack parallel to the axis of the bulb to relieve the stress, which is a typical hoop-stress failure. It could survive the assembly process, but fail prematurely in use, for a variety of reasons related to its high induced-stress levels. Or it might undergo stress relaxation sufficient to reduce the stress to a lower level that can be maintained.

Hoop-stress equations for two typical press-fit situations are shown in Fig. 8-72. The allowable design stress will depend on the particular plastic, temperature, and other environmental considerations. A simpler, though less accurate, method of evaluating press fits is to assume that the shaft will not deform when pressed into the hub. This is reasonably accurate when a metal shaft is used in a plastic hub. The hoop strain developed in the hub is then given by

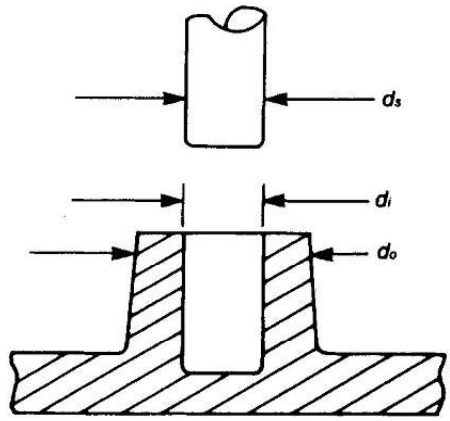
$$\varepsilon = \frac{i}{d_i}$$

The hoop stress can then be obtained by multiplying by the appropriate modulus. For high strains, the second modulus will give the initial stress; the apparent or creep modulus should be used for longer-term stresses. The main point is that the maximum strain or stress must be below the value that will produce creep rupture in the material. There is usually a weld line in the hub that can significantly affect the creep-rupture strength of most plastics.

An additional frequent complication with press fits is that a round hub or boss is often difficult to mold, if strict processing controls are not used to eliminate potential problems. There is a tendency for a round hub to be slightly elliptical in cross section, increasing the stresses on the part. For critical part performance and in view of what could occur, life testing should be conducted under actual conditions.

The consequences of stress occurring will depend on many factors, such as the temperature during and after assembly of the press

Spectrum Ex. 1014
IPR Petition - USP 11,536,632



E_p = MODULUS OF ELASTICITY OF PLASTIC
E_m = MODULUS OF ELASTICITY OF METAL
ν_p = POISSON'S RATIO OF PLASTIC
σ_a = ALLOWABLE DESIGN STRESS FOR PLASTIC
$i = d_s - d_i$ = DIAMETRAL INTERFERENCE
i_a = ALLOWABLE INTERFERENCE

CASE A
SHAFT AND HUB ARE BOTH THE SAME OR ESSENTIALLY SIMILAR MATERIALS
HOOP STRESS GIVEN "i" IS

$$\sigma = \frac{i}{d_s} E_p \frac{\Gamma}{\Gamma + 1}$$

OR, THE ALLOWABLE INTERFERENCE IS

$$i_a = d_s \cdot \frac{\sigma_a}{E_p} \frac{\Gamma + 1}{\Gamma}$$

GEOMETRY FACTOR

$$\Gamma = \frac{1 + \left(\frac{d_s}{d_o}\right)^2}{1 - \left(\frac{d_s}{d_o}\right)^2}$$

CASE B
SHAFT IS METAL, HUB IS PLASTIC
HOOP STRESS GIVEN "i" IS

$$\sigma = \frac{i}{d_s} E_p \frac{\Gamma}{\Gamma + \nu_p}$$

OR, THE ALLOWABLE INTERFERENCE IS

$$i_a = d_s \frac{\sigma_a}{E_p} \frac{\Gamma + \nu_p}{\Gamma}$$

Fig. 8-72 Press-fit conditions for two typical situations.

fit, modulus of the mating material, type of stress, service environment, and—probably most important—type of material being used. Some substances will creep or stress relax, but others will fracture or craze if the strain is too high. Except for light press fits, this type of assembly design can be risky enough for the novice, because the boss might already be weakened by a knit line. Figure 8-73 presents alternative methods for using press fits that present a lower risk of failure.

Internal Plastic Threads

The strength of plastic threads is limited, and when molded in a part involving either an unscrewing device or a rounded shape of thread—similar to bottle-cap threads—they can be stripped from the core. Screw threads, when needed, should be of the coarse type and have the outside of the thread rounded so as not to present a sharp V to the plastic, which can produce a notch effect.

If a self-threading screw can be substituted, it will not only appreciably decrease mold maintenance and mold cost but most likely, with proper type selection, also give better holding power. A screw that has a thin thread with relatively deep flights can give high holding power. If the screw or plastic is preheated to about 121°C (250°F), a condition of thermoforming in combination with material displacement will exist, thereby improving holding power. When male plastic threads are being considered the coarser threads are again preferred, and the root of the thread should be rounded to prevent the notch effect (Figs. 8-74 and 8-75).

External Plastic Threads

Threads can be molded or tapped into a plastic (Fig. 8-76). Molded internal threads usually require some type of unscrewing or collapsing mechanism. External threads can be molded by either splitting the mold

Spectrum Ex. 1014
IPR Petition - USP 11,536,632

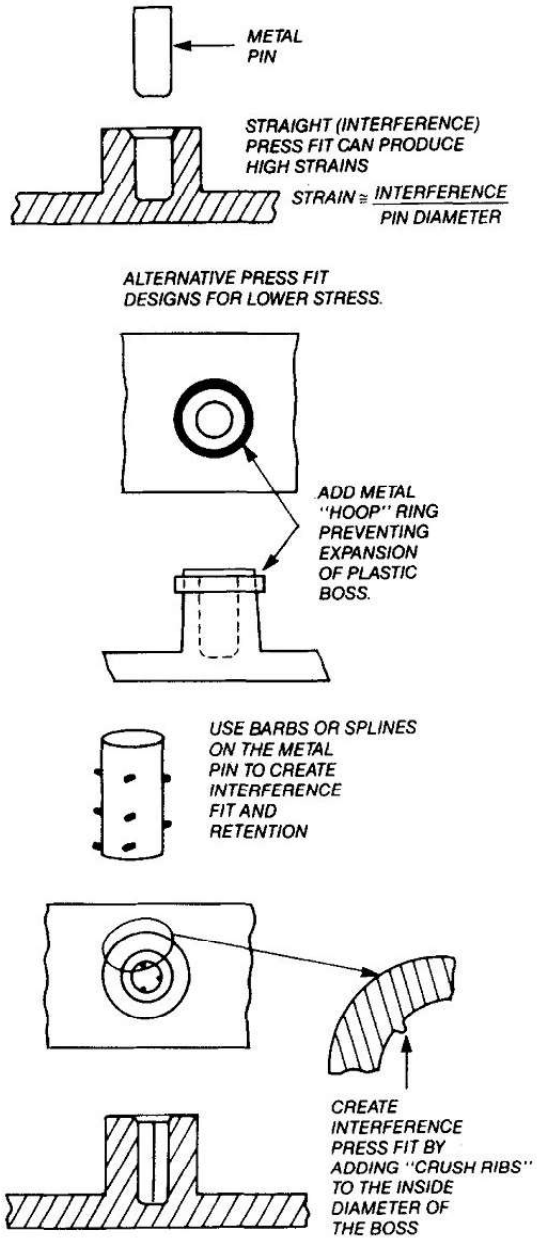


Fig. 8-73 Alternate press-fit designs for metal pins in a plastic hub.

halves or parting the line across the thread (Fig. 8-76), if parting the line on the threads is permitted. With a split mold, it is easier to design the mold and remove the threaded part from the mold during processing. The design of the threads requires control, to prevent excessive shear, resulting in stripping the threads when torqued, and also to limit hoop stresses, which can result in tensile failure. Although the mechanics of stress analysis for screw threads are readily available, the equations for them can be rather complicated. A simplified approximate diagram is presented here (Fig. 8-77).

Molded-In Inserts

If metal inserts are to be molded into a plastic product, their shape should present no sharp edges to the plastic, since the effect of the edges would be similar to that of a notch. A knurled insert should have the sharp point smoothed, again to avoid the notch effect. The practice of molding inserts in place is usually employed to provide good holding power for plastic products, but there are drawbacks to this method: It is dangerous to have an operator place an arm between the mold halves while the electrical power to the machine is turned on. It normally takes a pin to support the insert and since this pin is small in relation to the cored hole for the insert, it is easily bent or sheared under the influence of injection pressure. Should the insert fall out of position, there is danger of mold damage. Also, the hand placement of inserts contributes to cycle variation and with it potentially product quality degradation. Some

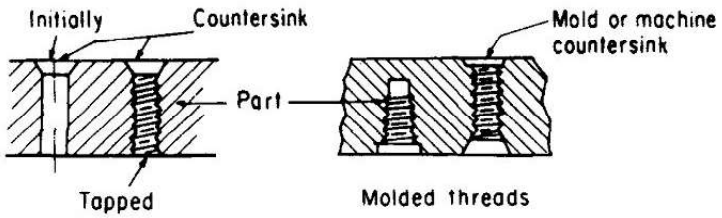


Fig. 8-74 Holes having molded-in threads or ones that are to receive tapped threads should have molded-in countersinks to avoid chipping and burrs. If a threaded pin is being used to mold threads in a through hole, the lead thread on the mold pin should have a pilot thread for support.

Spectrum Ex. 1014
IPR Petition - USP 11,536,632

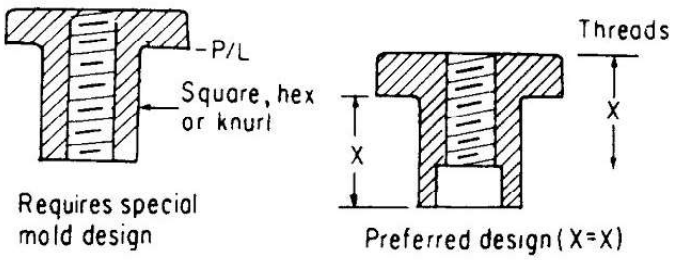


Fig. 8-75 Automatic unscrewing is expensive when molding, unless the plastic part is designed to prevent turning during the unscrewing cycle. When thread strength is a requirement, fine threads should not be molded, particularly in RPs where the threads could be mainly resin and thus very brittle. However, with proper molding procedures, based on type of compound or composite used and the design of the mold being gained through experience, no problem should develop.

of these problems can be overcome by higher mold expenditures, as, for example, shuttling cavities (see Figs. 8-78 through 8-91 and Table 8-4).

However, the desired results in fastening can be attained by other means, for example, by coring holes in the part to permit ultrasonic welding of inserts in place, by coring a hole in the part that will be of a size when the part is removed from the mold to permit a slight press fit plus a gain in the holding power from postmolding shrinkage, or by coring a hole in the part to permit dropping the insert and providing a retaining shoulder by spinning or ultrasonic forming.

All these assembly methods require the same time to perform as placing inserts in the mold, but they also lower machine time. There are probably several other means of accomplishing the desired result that depend on the circumstances at hand. In any event, molded-in inserts, in the long run, usually prove costlier and so should be avoided.

Screws for Mechanical Assembly

For mechanical assemblies using screws (Figs. 8-92 through 8-94), the screws can be detached indefinitely, with the exception of self-tapping screws, which can be loosened and retightened only a limited number of times. The best guideline for the designer is to prefer any assembly design that converts eventual tensile loads to compression loads. For those plastics that are subject to crazing or stress cracking, compression loads tend to reduce this problem. When feasible, use metal-to-metal force-locking connections, particularly with many of the TPs, to reduce stress on the plastic. The forces that can be applied with a single small screw can be surprisingly high. As in metals, consider the use of torque-limiting wrenches in designs where the degree of loading is critical.

External and internal threads can be molded economically in plastic parts (Fig. 8-92). Screw threads produced by the mold

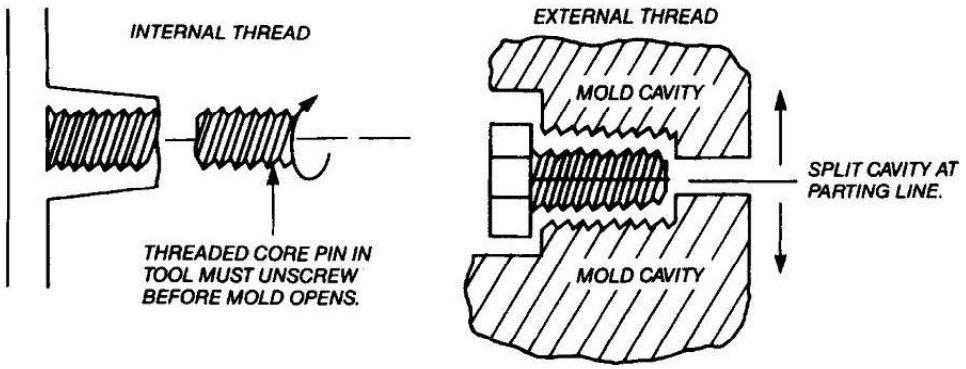


Fig. 8-76 Examples of internal and external threads.

Spectrum Ex. 1014
IPR Petition - USP 11,536,632

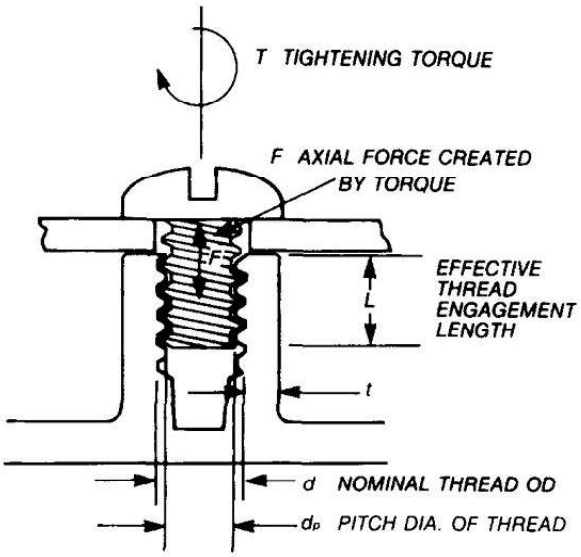


Fig. 8-77 The torque–force relationships of screw threads.

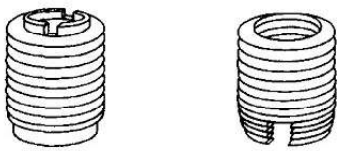
PUSH-IN TYPE INSERTS
ADVANTAGES — *SPEED AND LOW EQUIPMENT COST.*

DISADVANTAGES — *HIGH INDUCED STRESS AND ONLY FAIR HOLDING POWER.*



SELF-THREADING INSERTS
ADVANTAGE — *INSTALLED WITH MINIMAL EQUIPMENT.*

DISADVANTAGES — *SLOW AND THEY CAN CREATE HIGH STRESSES.*



EXPANSION TYPE INSERTS
ADVANTAGE — *NO INSTALLATION EQUIPMENT.*

DISADVANTAGES — *LOWER PERFORMANCE AND MODERATELY HIGH INDUCED STRESS.*

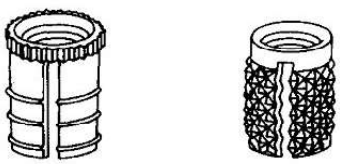


Fig. 8-78 Examples of common threads' metal inserts.

Spectrum Ex. 1014
IPR Petition - USP 11,536,632

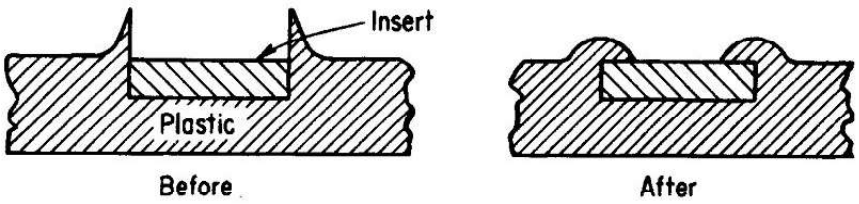


Fig. 8-79 The technique of using hot-roll plastic over inserts to obtain strengthened anchorages.

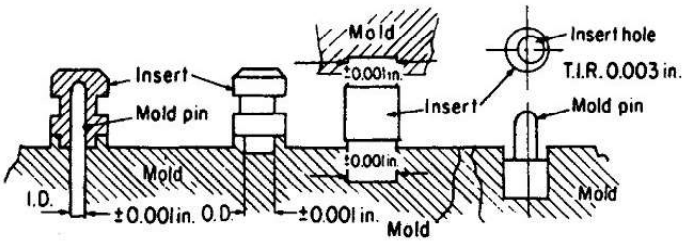


Fig. 8-80 Either inside or outside diameters can be used to hold inserts in place during molding. Tight tolerance [±0.001 in. (0.0025 cm) with a maximum TIR of 0.003 in. (0.0076 cm)] is required.

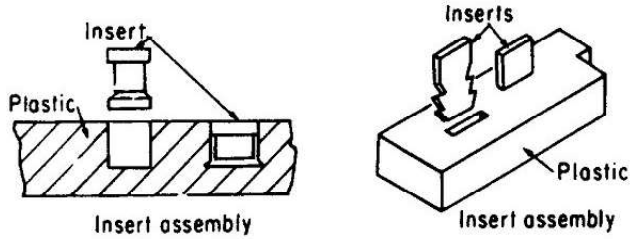


Fig. 8-81 Assembling inserts after molding prevents problems such as potential melt flow over metal surfaces during molding and the scratching of plated inserts during any required deflashing. Pressed-in inserts require holes sized for proper fits.

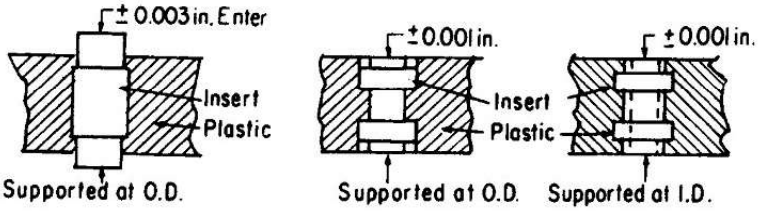


Fig. 8-82 Generally, through inserts must be molded ±0.001 in. (0.0025 cm) of length to ensure their making contact with both mold surfaces.

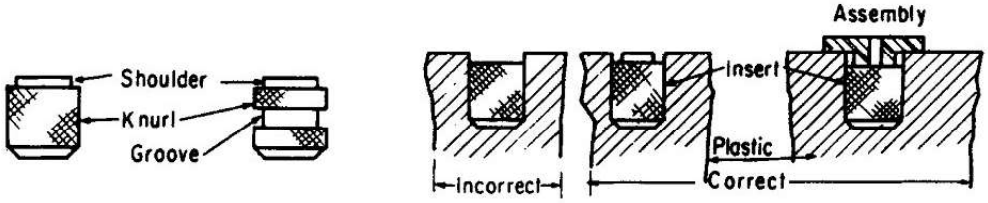


Fig. 8-83 Knurls and grooves can serve to anchor inserts in molded products. They should be flush with the top surface or in contact with an assembling member to prevent their jacking out.

Spectrum Ex. 1014
IPR Petition - USP 11,536,632

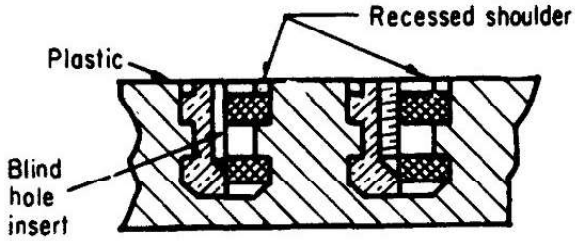


Fig. 8-84 During molding, any loose steel inserts falling into the molds will damage their cavities, but brass or soft metals such as aluminum will usually be crushed, with minimal or no damage to the mold. The flow of plastic into the interior of an insert is impeded by using a blind-hole insert and shoulder [about $\frac{1}{32} \times \frac{1}{32}$ in. (0.08 × 0.08 cm)] around the insert opening.

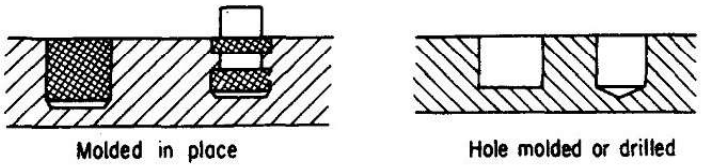


Fig. 8-85 Some of the methods of assembling inserts include mechanical pushing and sonic pushing, or allowing hot plastic to shrink around the insert.

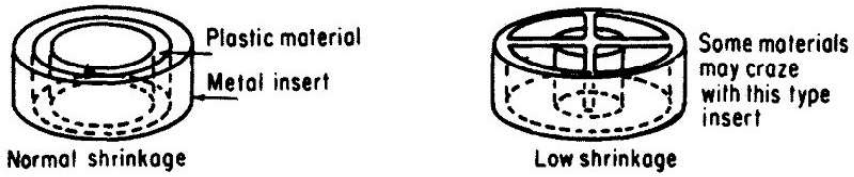


Fig. 8-86 Plastic generally shrinks away from metal when it is molded inside a metal insert. Both the insert's structure and type of plastic used will determine the amount of shrinkage.

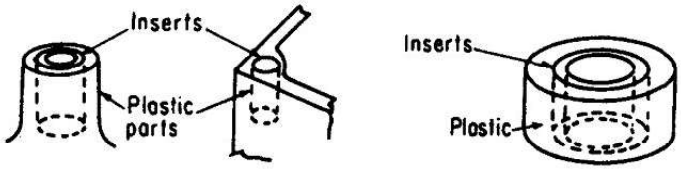


Fig. 8-87 Metal stamping and inserts of various shapes are usuable in many ways. To prevent cracking and crazing during aging under use, surround all inserts with reasonably thick plastic walls. Thin walls can crack and too-thin walls can also show sink marks.

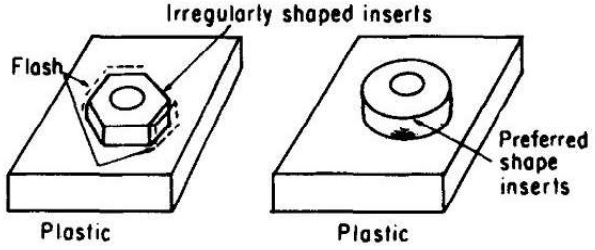


Fig. 8-88 Irregularly shaped inserts protruding from the plastic complicate mold construction, adding costly deflashing operations to the product.

Spectrum Ex. 1014
IPR Petition - USP 11,536,632

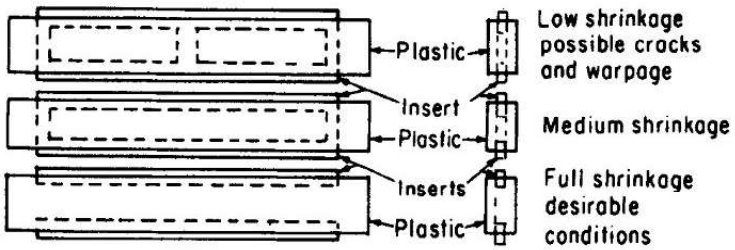


Fig. 8-89 When using metal embedments, the dimensions and tolerances of the plastic parts will be difficult to estimate, because metal tends to prevent plastic from shrinkage. The insert structure influences the final dimensions by controlling the movement of the plastic during aging, setting up stresses that can cause cracks and warpage.

itself using rotating cores, split inserts, or collapsible cores will eliminate the normally expensive postmolding threading operations. Because coarse threads are more easily molded than fine ones, threads less than 32-pitch should be avoided. American Standard screw threads should be designed and molded carefully. If the threads end up forming notches, a reduction in impact strength and ultimate elongation under tensile stress can be significant, depending on the type of plastic used. With certain applications and materials, trapezoidal and knuckle threads are better.

Generally, the length of thread used should be more than 1.5 times the diameter, with the section thickness around the hole more than 0.6 times the diameter. Avoid having feather edges and limit tightening with the bolt shoulder.

Bottle caps made from different plastics are extensively used. Some closures are of the simple cork type, but most are of the screw type. Strong, accurate threads can be molded, which represent undercuts. Simple designs, such as wide-pitch threads, should be used when permitted. The thread should be designed to start about $\frac{1}{32}$ in. from the end of the face perpendicular to the axis of the thread. It is usually practical to mold up to 32 threads per inch; more than this number can give certain molders trouble.

Self-threading screws are an economical means of securing separable plastic joints. They can be either thread-cutting or thread-forming (Fig. 8-93). To select the correct self-threading screw, the designer should know which plastic will be used and its mechanical properties, particularly its modulus of elasticity. These self-threading screws are driven into the molded part, eliminating the need for a molded-in thread or secondary tapping operation. They differ in their thread spacing and body design. Thread-forming screws, which provide the highest stripping torques, have less tendency to damage threads in repeated assembly operations than do other types. The thread-forming screw displaces material as it is being installed in the receiving hole. This type of screw induces high stress levels in the plastic part, so it is not recommended for use with certain plastics

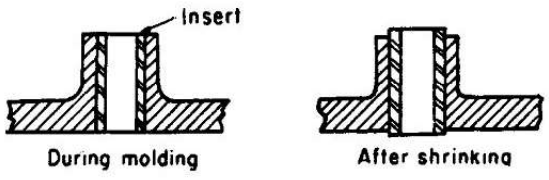


Fig. 8-90 Molded materials with high shrinkage values during cooling will shrink below the metal inserts, allowing the insert ends to extend beyond the part.

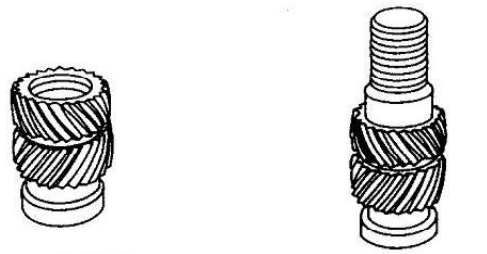


Fig. 8-91 These examples of ultrasonic inserts, designed for excellent performance, result in fast action with very little induced stress.

Spectrum Ex. 1014
IPR Petition - USP 11,536,632

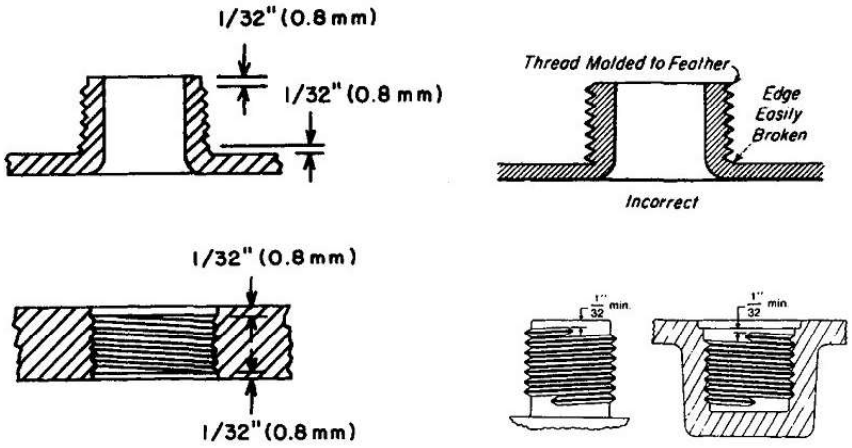


Fig. 8-92 Thread design guides.

such as those with a low modulus, unless careful procedures are used in forming the threads.

Screws or threaded bolts with nuts require through-going holes but provide an easy assembly system. As shown in Fig. 8-94, washers are recommended to distribute the load on a larger area, wherever feasible or required. If a screw is tightened too far, excessive bending or tensile stresses will easily be created, possibly causing cracking based on stress-to-failure data curves. A change in design or the use of a spacer can convert tensile into compressive stresses. Different screw-and bolt-heads can be used, but flat-underside types of heads are best.

Gears

Gear design is one of the more complicated areas for designing with plastics, because the bending, shear, rolling, and sliding stresses all act on a mechanism whose purpose is to transmit uniform motion and power. In this age of light weight and quieter operation, plastic gears have become increasingly important as a means of cutting cost, weight, and noise without significantly reducing performance. Because plastics are not as strong as steel, they must perform far closer to their design limits than do metal gears. Although many plastic gear designs are derived from metal-gear technology, plastics demand special consideration, for instance, to deal with heat buildup from hysteresis.

The basic difference between metal and plastic in gear design is that designs for metal are based on the strength of a single tooth, whereas plastic shares the load among the various gear teeth to spread it out. Thus, in plastics the allowable stress for a specific

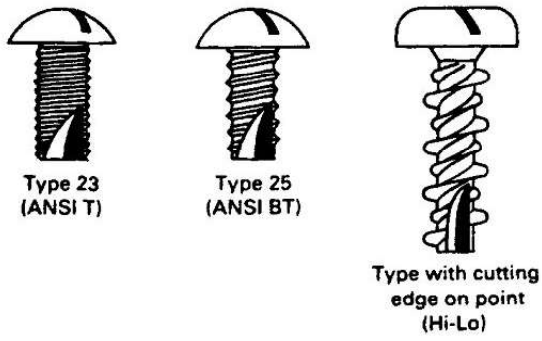


Fig. 8-93 Examples of self-threading screws.

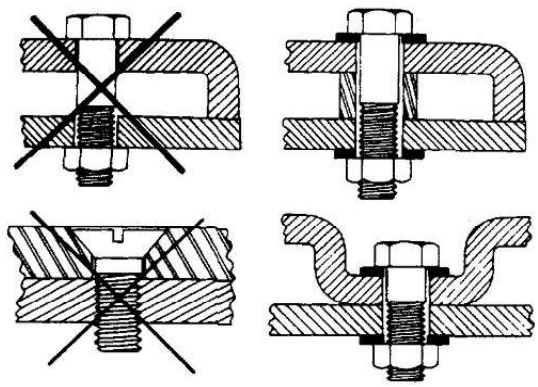


Fig. 8-94 Examples of screw and bolt assemblies with the better designs on the right side.

Spectrum Ex. 1014
IPR Petition - USP 11,536,632

number of cycles to failure increases as the tooth size decreases, to a pitch of about 48. Very little increase is seen above a 48 pitch, because of the effects of size and other considerations. The following guidelines for good gear design with TPs should be observed: (1) Determine the gears' conditions of service, such as temperature, load, velocity, space, and environment; (2) establish the short-term plastic properties as against the initial performance requirements; (3) compare the long-term property retention factor as opposed to the life of the gear; (4) using physical property data, calculate the stress levels caused by the various loads and speeds; and (5) then compare these calculated values with the allowable stress levels and redesign as needed to provide an adequate safety factor (Figs 8-8 and 8-9).

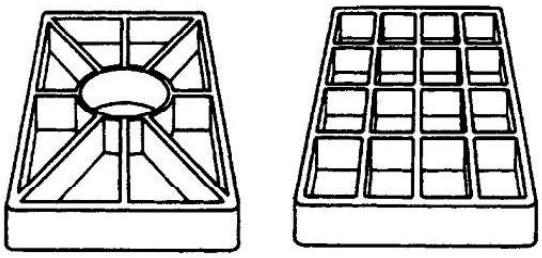


Fig. 8-95 Example of a guideline in ribbing: Bidirectional ribbing (right) can reduce or eliminate sagging or bending; circular or diagonal ribbing (left) reduces or eliminates twisting.

Ribs

If there is sufficient space, the use of ribs is a practical, economic means of increasing the structural integrity of plastic parts without creating thick walls (see Chap. 5). Ribs are provided for spacing purposes, to support components, and for other uses. Table 8-1 gives a summary of the results of using a rib design.

Although the use of ribs gives the designer great latitude in efficiently tailoring the structural response of a plastic part, ribbing can result in warping and appearance problems. In general, experienced design engineers do not use ribs if there is doubt as to whether they are structurally necessary. Adding ribs after the tool is built is usually simple and relatively inexpensive since it involves removing steel.

There are certain basic rib-design guidelines that should be followed (Figs. 8-95 and 8-96). The most general is to make the rib thickness at its base equal to one-half the adjacent wall's thickness. With the ribs' opposite appearance areas, the width should be kept as thin as possible. In areas where structure is more important than appearance or with very low-shrinkage materials, ribs are often 75 or even 100% of the outside wall's thickness. As can be seen in Fig. 8-96, a goal in rib design is to prevent the formation of a heavy mass of material that can result in a sink, void, distortion, long cycle time, or any combination of these problems.

All ribs should have a minimum of $\frac{1}{2}$ deg of draft per side and minimum radius of 0.005 in. at the base. Generally, the draft and thickness requirements will limit the height of the rib. Therefore, multiple, evenly spaced ribs are preferred to single, large ribs. Whenever possible, ribs should be smoothly connected to other structural features such as bosses, side walls, and component-mounting pads. Ribs need not be constant in height or width and are often matched to the stress distribution in the part.

The first step in designing a rib is to determine the dimensional limitations, followed by establishing what shape the rib will have to be to realize a part with good strength and satisfactory appearance that can be produced economically. Figure 8-97 shows the proportional dimensions of rib versus thickness. This arrangement will minimize voids (sinks), stresses, and shrinkage variations and lends itself to trouble-free molding.

Poor heat conductivity during molding creates temperature gradients throughout the cross section, with resultant stresses. Thus heavy walls will have inferior properties. Cycle times are usually exceptionally long, another cause of stress. Also, close-tolerance dimensions are difficult to maintain, material is wasted, quality degraded, and cost increased. If performance calculations indicate wall thicknesses well above those recommended for a particular material, one solution is to achieve equivalent cross-sectional properties

Spectrum Ex. 1014
IPR Petition - USP 11,536,632

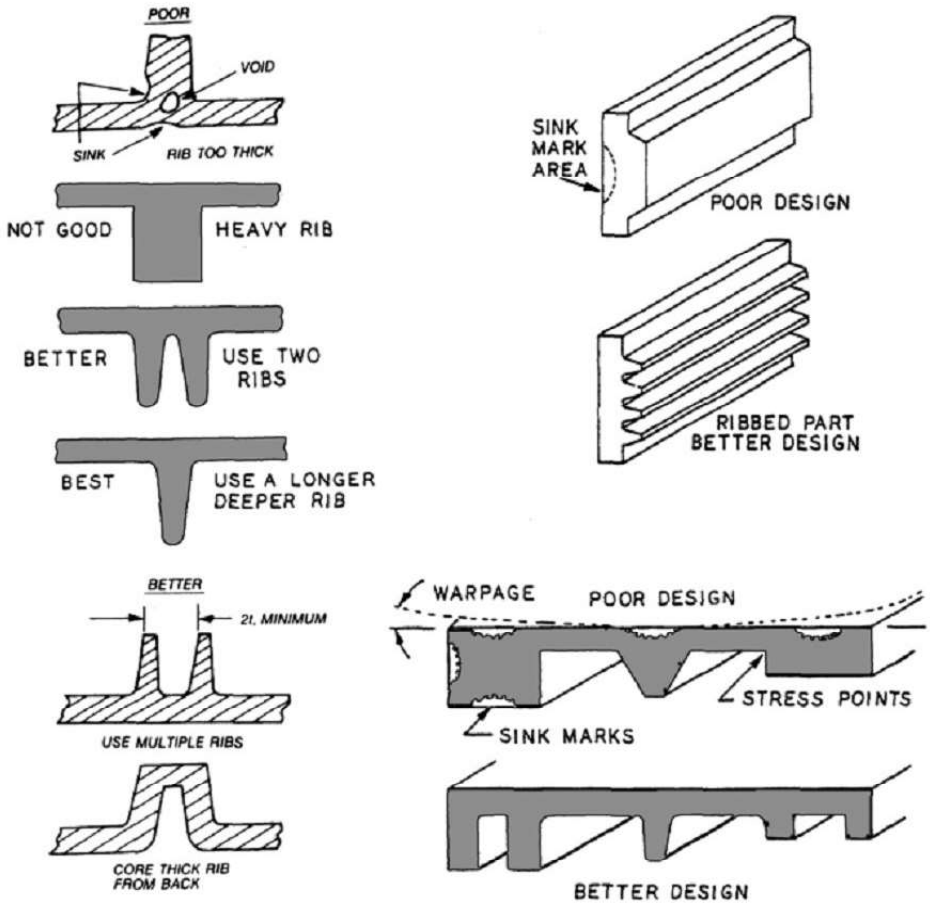


Fig. 8-96 Examples of rib design characteristics.

by ribbing. Solid plastic wall thicknesses for most materials should be below 0.2 in., preferably around 0.125 in., in the interest of avoiding these pitfalls. In most cases, ribbing will provide a satisfactory solution; in others, reinforced material may have to be considered.

An example of how ribbing can provide the necessary equivalent moment of inertia and section modulus follows.

A flat plastic bar $1\frac{1}{2}$ in. wide, $\frac{3}{8}$ in. thick, and 10 in. long, supported at both ends and loaded at the center, was calculated to provide a specified deflection and stress level under a given load. The favorable material thickness of this plastic is 0.150 in., and its rib proportions would be as in Fig. 8-97.

If we use judgement as a guide, it would appear that the $1\frac{1}{2}$-in. width would require about two ribs. So, as a starting point, let us calculate the equivalent cross-sectional data as if we were dealing with two T sections.

According to standard engineering handbooks (under "Stress and Deflection in Beams," "Moments of Inertia," etc.), resistance to stress is expressed by the moment of inertia and the resistance to deflection by the section modulus. By finding a cross section with the two factors equivalent, we can assure equal or better performance in the ribbed design compared to the thick wall without ribs.

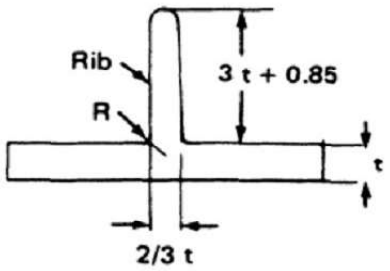


Fig. 8-97 Rib proportion versus thickness.

The stress $= Wl/4Z$ and deflection $= Wl^2/48EI$

where W = load
l = length of beam
Z = section modulus
E = modulus of elasticity
and I = moment of inertia

For the flat bar $I = bd^3/12$, $b = 1\frac{1}{2}$, $d = \frac{3}{8}$; or $I = (1.5 \times 0.375^3)/12 = 0.0066$ for the rectangular bar. The section modulus is

$$Z = \frac{L}{Y} = \frac{0.0066}{0.1875} = 0.0352$$

The moment of inertia of a T section is

$$I = \tfrac{1}{12}[4bs^3 + h^3(3t + T)] - A(d - y - s)^2$$

where $b = 0.75$
$s = 0.15$
$h = 0.6$
$t = 0.8$
$T = 0.1$
$d = 0.75$
A = area $= bs + h(T + t)/2$

Therefore,

$$y = d - [3bs^2 + 3ht(d + s) + l(T - t)(h + 3s)]/6A$$

where y is the distance from the neutral point to the extreme fiber. Substituting the values in the formulas, we have

$$A = (0.75 \times 0.15) + \frac{0.6(0.1 + 0.08)}{2} = 0.1665$$

$$y = 0.75 - \{(3 \times 0.75 \times 0.15^2) + [3 \times 0.6 \times 0.08(0.75 + 0.15)] + 0.6(0.10 - 0.08) \times (0.6 + 0.45)\}/(6 \times 0.1665)$$

$$= 0.75 - \frac{0.0506 + 0.1296 + 0.0126}{0.999} = 0.75 - 0.193 = 0.557$$

$$I = \tfrac{1}{12}\{(4 \times 0.75 \times 0.15^3) + 0.6^3[(3 \times 0.08) + 0.1]\} - 0.1665(0.75 - 0.557 - 0.15)^2$$

$$= \tfrac{1}{12}(0.0102 + 0.073) - 0.1665 \times 0.00185 = 0.00693 - 0.00031 = 0.00662$$

$$Z = \frac{0.00662}{0.557} = 0.0119$$

Two of the T sections would provide a higher moment of inertia and decreased section modulus than a sandwich structure. When placed on the end, the two ribs would

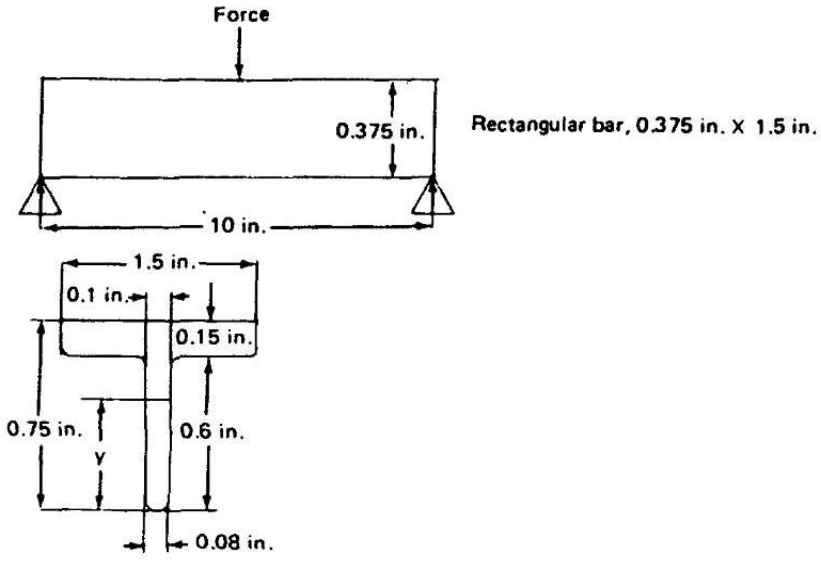


Fig. 8-98 Single-rib shape with good moldability that gives a section modulus and moment of inertia equivalent to a rectangular bar (3, 4).

Spectrum Ex. 1014
IPR Petition - USP 11,536,632

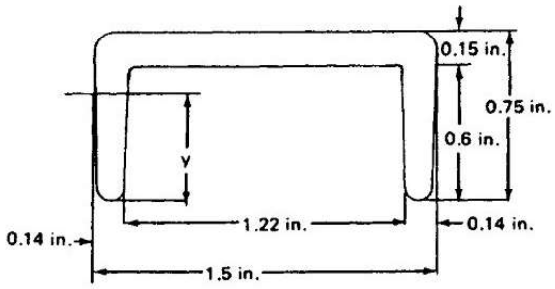


Fig. 8-99 Two-rib shape with good moldability that gives a section modulus and moment of inertia equivalent to a rectangular bar.

make a channel that would give a moment of inertia of 0.018 and section modulus of 0.035, values that are more than adequate for the purpose (Fig. 8-98). It should be noted that the two-rib construction forming a channel would require only 70% of the material used in a solid bar (Fig. 8-99).

Other means of stiffening surfaces can also be used, if appearance permits, where areas can be domed and corrugated. The basic goal in any action that leads to greater rigidity is to specify a practical wall thickness that will optimize strength and processing and thus result in high-quality products. In addition to ribs, other features protruding from a wall, such as bosses or tabular shapes, should be treated similarly as far as transition radius, taper, and minimal material usage are concerned. The same principles are involved. Identical ill effects can be expected unless the recommended practices are incorporated.

Geometric Structural Reinforcement

Besides ribs, there are other methods of improving section properties. Many of these can often be designed into functional or appearance features for the product. Basically, to increase load-carrying ability or stiffness, it is necessary to increase either the properties of the plastic material or section properties of the structure. Beneficial improvements in the material can made in certain cases by changing the grade or type, compounding or alloying, and incorporating fillers or fiber reinforcements (Chap. 6). As far as structure is concerned, thickening wall sections will often prove a practical means of increasing section properties, but there can certainly be economic limitations in terms of material usage and molding cycles. Some typical examples of these structures and potential problems to consider are shown in Figs. 8-100 through 8-102.

Some geometric shapes that provide the designer with alternate methods of increasing

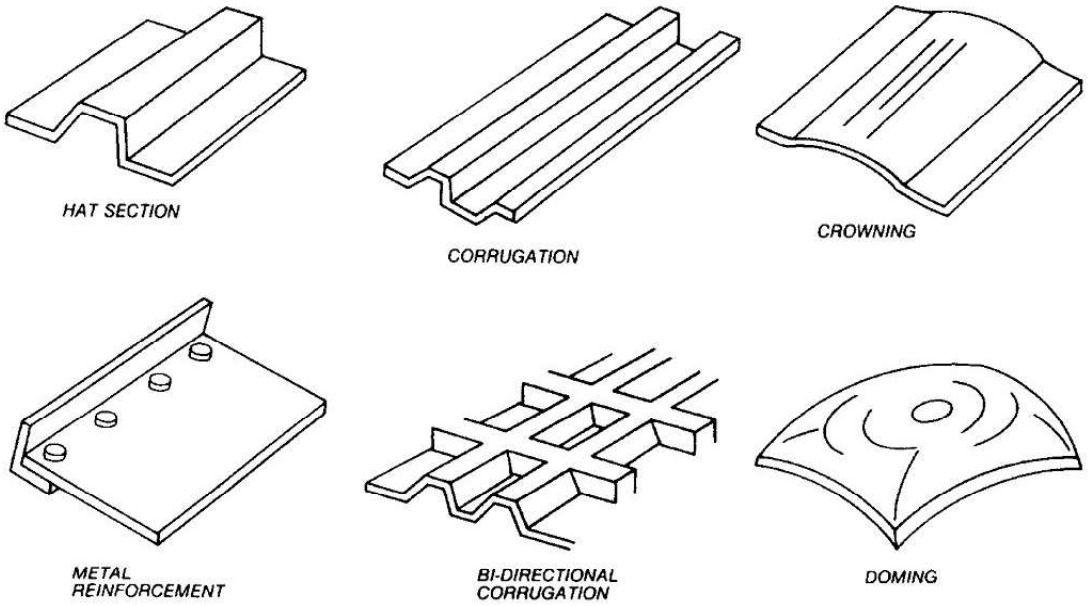


Fig. 8-100 Examples of geometric structural reinforcement techniques.

Spectrum Ex. 1014
IPR Petition - USP 11,536,632

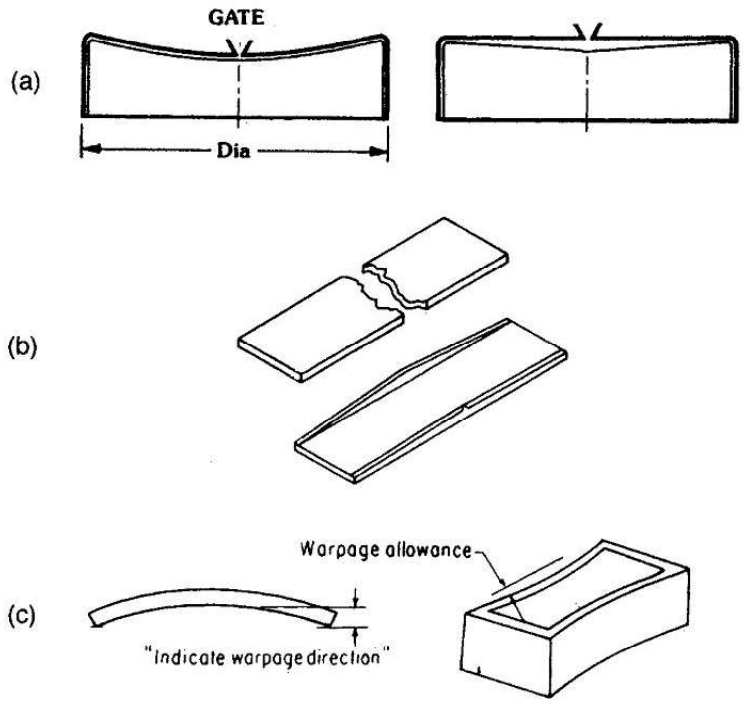


Fig. 8-101 Examples of potential problems with different shapes. (a) Oil canning effect; left view is with even wall; right shows redesigned double-tapered wall. (b) Long, flat part (top) could tend to warp; add ribs to correct problem. (c) Warpage allowance and direction should be shown on the design drawing so that cooling fixtures, type of plastic, and molding techniques may be determined to achieve optimum performance in the molded part.

part supports include gussets, corrugating, and doming. Gussets are supporting structures for either the edge of a part or bosses. The design guidelines for gusset thicknesses, spacing, and taper or draft are the same as for ribs. The dimensions given in Fig. 8-102 for height, length, and spacing are the lowest acceptable dimensions for good gusset design.

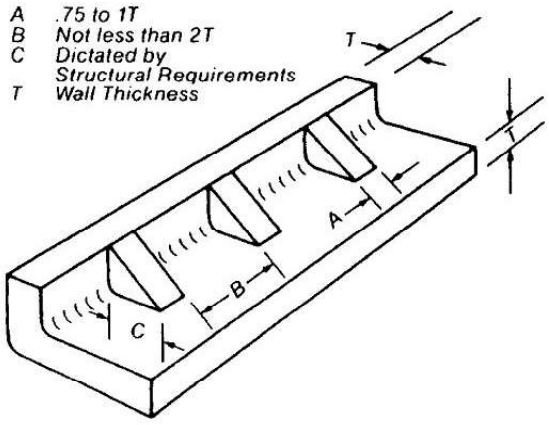


Fig. 8-102 Guidelines for designing a gusset.

Corrugating and doming provide the designer increased part performance without having to add ribs. Of these two methods, corrugating is the more effective. The alternative to corrugating is doming (Fig. 8-100). When a part is domed, its wall is molded in a convex shape. In some cases, this method may be preferred to corrugating for aesthetic purposes. From a structural stand-point, though, doming does not offer the same reinforcement rigidity of corrugating.

Snap Joints

A snap joint is economical in two respects: It allows the structural member to be molded simultaneously with the molded part, and it allows rationalizing the assembly, compared with such other joining processes as screws. Table 8-5 provides a comparison of its advantages and disadvantages. The design guidelines that should be taken into consideration

Spectrum Ex. 1014
IPR Petition - USP 11,536,632

Table 8-5 Advantages and disadvantages of snap joints

Advantages	Disadvantages
Can be easily integrated into the structural member.	The fixing of the joined parts is weaker than in welding, bonding, and screw joining.
Compact, space-saving form.	The conduct of force at the joining place is less than in a real joining (bonding, welding).
Takes over other functions like bearing, spring cushioning, fixing.	Effects of processing on the properties of the snap joints (orientation of the molecules and filler, distribution of the filler, binding seams, shrinkage, surface roughness and structure).
Higher forces can also be transmitted with proper designing.	Narrow tolerances are required in complicated applications (in plastics, this is associated in some cases with considerable expenditure).
Small number of individual parts.	Influence of environmental effects (e.g., distortion due to temperature differences) on the functioning.
Assembly of a construction system with little expenditure of production facilities and time.	Difficulties with a continuous loading of the snap joint.

for obtaining the desired functions are described in Chap. 5. Some examples of the various types and their design considerations are shown in Figs. 8-103 through 8-105.

The geometry for snap joints should be chosen in such a manner that excessive increases in stress do not occur (see Chap. 5). The arrangement of the undercut should be chosen in such a manner that deformations of the molded part from shrinkage, distortion, unilateral heating, and loading do not disturb its functioning. The following guidelines are recommended regarding the position of the snap joint to the gate and choice of the wall thicknesses in the area of flow to the place of joining: (1) There should be no binding seams at critical points; (2) avoid binding seams created by stagnation of the melt during filling; (3) the molecules and filler should be oriented in the direction of stress; and (4) any uneven distribution of the filler should not occur at high-stress points.

The stressing and joining forces should not act in the same direction if the joint has to absorb larger forces. Arm brackets should possibly be used to prevent the hook from escaping after joining. When there is a fracture of the snap hook as a result of overloading during the joining operation, the cross section should not be increased, but the hook should be designed to be more flexible. On account of the frictional forces and stresses that appear at the point of joining, the angle of joining should be chosen to be not larger than 60 deg.

Integral Hinges

Hinge dimensions for lids, boxes, caps, and many other products have by now been well established (see Chap. 5). Figure 8-106 shows the successful dimensions of a living hinge. Not only are the dimensions dictated by the design shape, they must also ensure that during the molding operation melt flow through the hinge is perpendicular to the hinge's bending action so that the plastic molecules stretch to give a strong, pliable hinging section. Generally, to ensure proper hinge action, the hinge is flexed immediately after the part is molded while it is still hot, using mechanical action in the mold or flexing it manually. It is important to locate gates in the proper position in relationship to the thickness and flow pattern of the melt so that the melt flows properly through the hinge. For example, a poor flow condition would exist if the gates were on opposite sides of a hinge. This would form a weld line within the hinge, causing it to fail upon its first being bent.

Some examples of a few of the thousands of successful living hinge applications are shown

Spectrum Ex. 1014
IPR Petition - USP 11,536,632

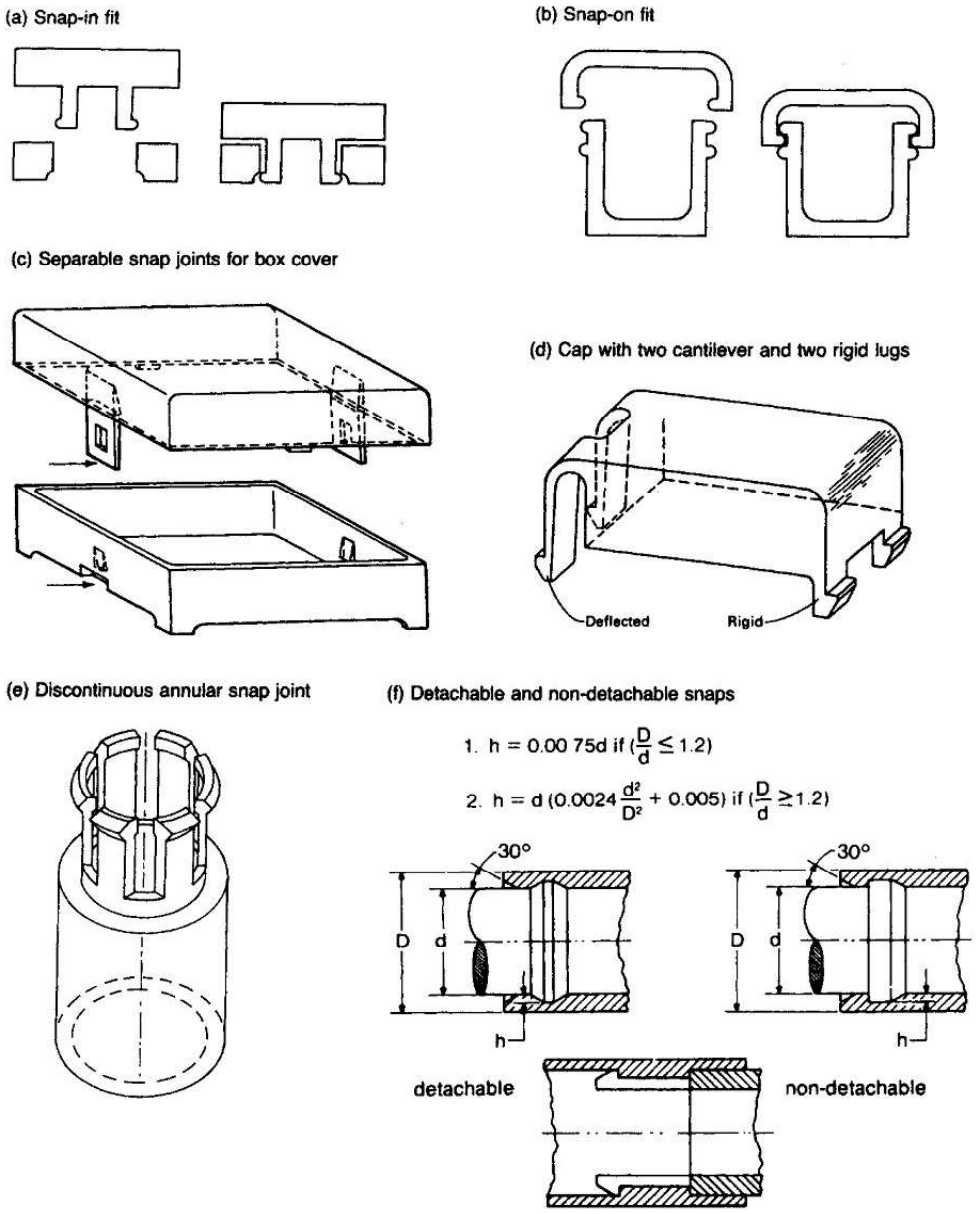


Fig. 8-103 Examples of different snap-fit designs. (a) Snap-in fit. (b) Snap-on fit. (c) Separable snap joints for box cover. (d) Cap with two cantilever and two rigid lugs. (e) Discontinuous annular snap joint. (f) Detachable and nondetachable snaps.

in Fig. 8-107. Others include dual-flap cap closures, lifting cap tabs, and diaphragm valve flex hinges.

Mold Action

Molds can be designed to produce products that permit molding from very simple shapes to extremely complex ones. The complex shapes can include practically all those actions reviewed in this section that have been classified as being poor or difficult to mold. In a simple mold design, which is defined as when the mold separates into two or more sections, the part can be removed. For this to be done, certain geometric considerations must be met. First, the part should have no undercut sections that will lock if it is pulled from the mold.

Spectrum Ex. 1014
IPR Petition - USP 11,536,632

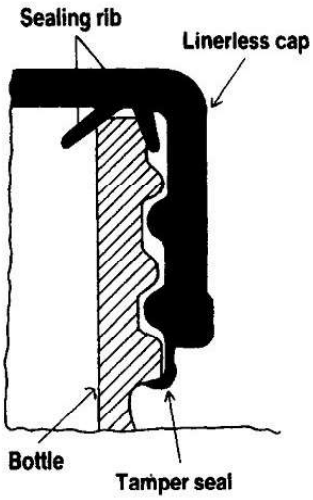


Fig. 8-104 Section of a linerless cap with a tamper-proof seal.

Cases in which shape is essential to function, require a more elaborate mold, where a portion is retracted to permit the undercut to be removed. This complicates the molding procedure and mold, which 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 (the parting line), and pulling the part against these parallel surfaces could result in sticking and drag, which 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

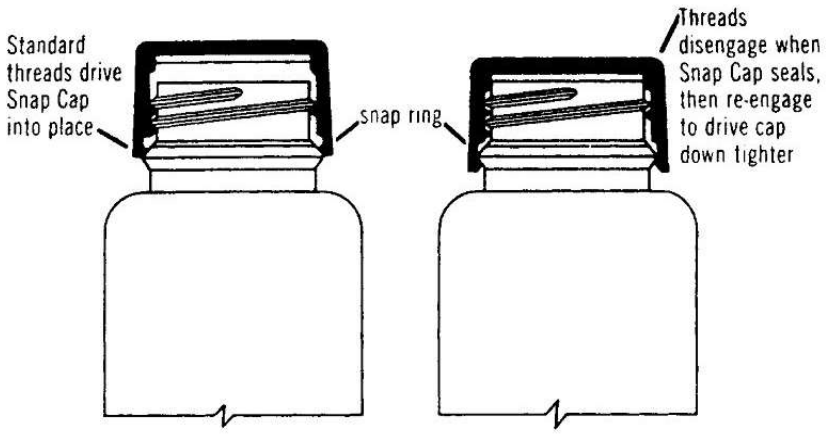


Fig. 8-105 This nonbackoff cap, or NBO, goes on like a screw cap and seals like a snap cap to provide a liquid-tight closure.

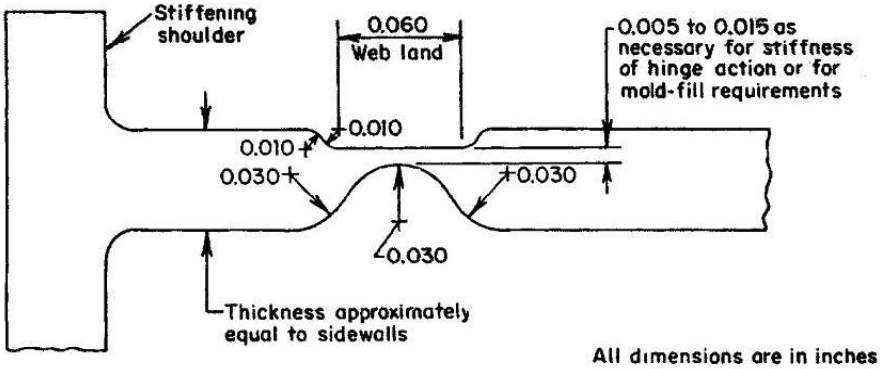
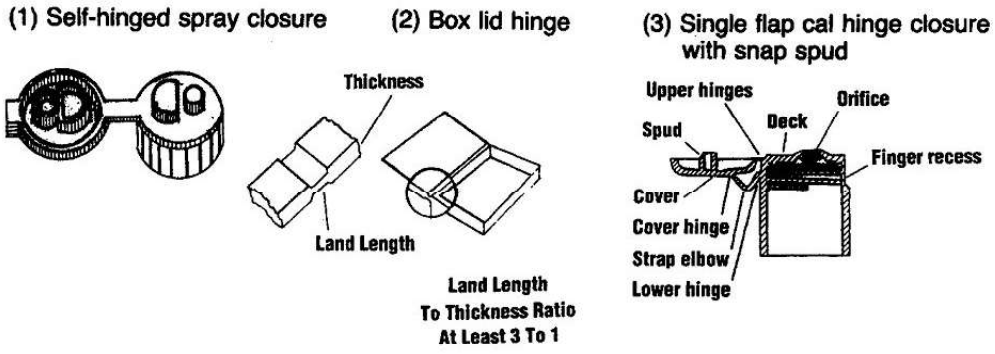


Fig. 8-106 Example of basic polypropylene integral hinge design where the plastic melt flow in a one-way direction is from the thick section through the thin section of the hinge. This melt flow through the thin section causes a degree of molecular orientation that is directly related to its performance as a hinge. (All dimensions are in inches.)

Spectrum Ex. 1014
IPR Petition - USP 11,536,632



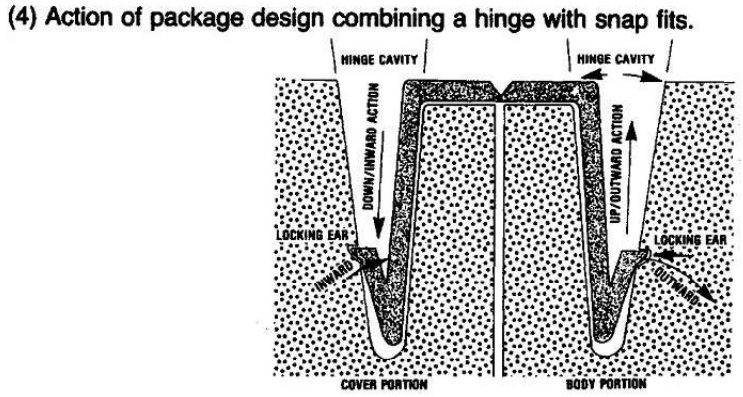


Fig. 8-107 Some examples of living hinges.

design. Moldings made from flexible plastic with small undercuts often allow forced mold release; that is, during the mold opening, the molding distorts sufficiently, because of its flexibility, to jump free of the undercut. This method is not recommended for the novice. In such cases, a certain degree of deformation may have to be accepted. Generously rounded corners are a must if this method of mold release is to be used.

For rigid plastics and large undercuts, use must be made of movable or rotating side cores, which then obviously influence mold constructions. 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. 8-108).

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, peumatically, or electromechanically:

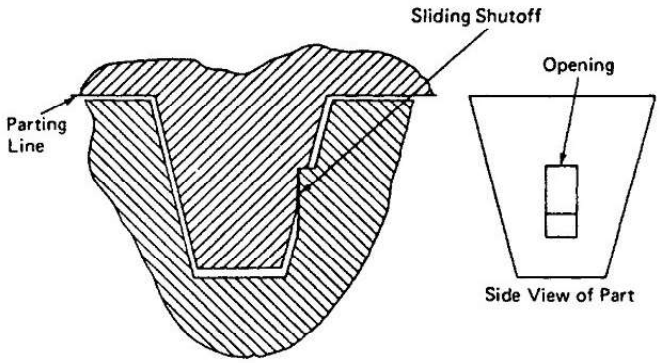


Fig. 8-108 Eliminate an undercut, requiring side cores, by using a sliding shutoff system.

Spectrum Ex. 1014
IPR Petition - USP 11,536,632

Molds with side cores (Figs 4-118 through 4-120)

Molds with wedges (Figs. 4-120 and 4-121)

Molds with rotating cores (Figs. 4-121 and 4-122)

Molds with loose cores or inserts (Fig. 4-83)

The choice of method, or a combination of these methods, is governed by not only the shape of the article and properties of the polymer (its flexibility, rigidity, shrinkage, etc.) but also the standards of quality to be met by the article. For articles with an external screw thread, for instance, either the first or third method can be used. However, if the first is used, the mold parting line shows, which may be undesirable for aesthetic or design reasons. The method used should depend entirely on the circumstances of the article.

Spectrum Ex. 1014
IPR Petition - USP 11,536,632