Sample Pages

David O. Kazmer

Injection Mold Design Engineering

Book ISBN: 978-1-56990-570-8


For further information and order see

www.hanserpublications.com (in the Americas)

www.hanser-fachbuch.de (outside the Americas)
Preface to the 2nd Edition .................................................. V
Preface to the 1st Edition .................................................... VIII

Nomenclature ................................................................. XV

1 Introduction ................................................................. 1
  1.1 Overview of the Injection Molding Process ....................... 1
  1.2 Mold Functions ....................................................... 4
  1.3 Mold Structures ...................................................... 6
    1.3.1 External View of Mold ......................................... 6
    1.3.2 View of Mold during Part Ejection ........................ 8
    1.3.3 Mold Cross-Section and Function .......................... 9
  1.4 Other Common Mold Types ........................................ 11
    1.4.1 Three-Plate, Multicavity Family Mold ....................... 12
    1.4.2 Hot Runner, Multigated, Single Cavity Mold ............... 14
    1.4.3 Comparison ..................................................... 15
  1.5 The Mold Development Process ................................... 16
  1.6 Mold Standards ..................................................... 18
  1.7 Chapter Review ..................................................... 20
  1.8 References .......................................................... 20

2 Plastic Part Design ...................................................... 21
  2.1 The Product Development Process ................................ 21
    2.1.1 Product Definition ........................................... 22
    2.1.2 Product Design ................................................ 23
    2.1.3 Development ................................................... 23
    2.1.4 Scale-Up and Launch ......................................... 24
    2.1.5 Role of Mold Design ......................................... 24
4  Mold Layout Design .................................................. 79
  4.1 Parting Plane Design .............................................. 79
    4.1.1 Determine Mold Opening Direction ...................... 80
    4.1.2 Determine Parting Line .................................. 83
    4.1.3 Parting Plane ............................................. 84
    4.1.4 Shut-Offs ................................................. 86
  4.2 Cavity and Core Insert Creation ............................... 87
    4.2.1 Height Dimension ....................................... 87
    4.2.2 Length and Width Dimensions ............................. 88
    4.2.3 Adjustments .............................................. 89
  4.3 Mold Base Selection ............................................. 91
    4.3.1 Cavity Layouts .......................................... 91
    4.3.2 Mold Base Sizing ....................................... 93
    4.3.3 Molding Machine Compatibility .......................... 95
    4.3.4 Mold Base Suppliers .................................... 97
  4.4 Material Selection ............................................... 98
    4.4.1 Strength vs. Heat Transfer ............................... 100
    4.4.2 Hardness vs. Machinability .............................. 101
    4.4.3 Material Summary ....................................... 102
    4.4.4 Surface Treatments ..................................... 103
  4.5 Chapter Review .................................................. 105
  4.6 References ..................................................... 107

5  Cavity Filling Analysis and Design ................................. 109
  5.1 Overview .......................................................... 109
  5.2 Objectives in Cavity Filling .................................... 110
    5.2.1 Complete Filling of Mold Cavities ....................... 110
    5.2.2 Avoid Uneven Filling or Over-Packing ................... 111
    5.2.3 Control the Melt Flow ................................... 112
  5.3 Viscous Flow ..................................................... 112
    5.3.1 Shear Stress, Shear Rate, and Viscosity ................. 112
    5.3.2 Pressure Drop ............................................ 113
    5.3.3 Rheological Behavior ..................................... 115
    5.3.4 Newtonian Model ........................................ 117
    5.3.5 Power Law Model ......................................... 119
  5.4 Process Simulation ............................................... 121
  5.5 Cavity Filling Analyses and Designs ........................... 124
    5.5.1 Estimating the Processing Conditions .................... 124
    5.5.2 Estimating the Filling Pressure and Minimum Wall Thickness ... 127
Contents

9 Cooling System Design ................................................. 243
  9.1 Objectives in Cooling System Design ............................ 243
    9.1.1 Maximize Heat Transfer Rates .............................. 243
    9.1.2 Maintain Uniform Wall Temperature ....................... 244
    9.1.3 Minimize Mold Cost ....................................... 244
    9.1.4 Minimize Volume and Complexity ......................... 245
    9.1.5 Maximize Reliability ..................................... 245
    9.1.6 Facilitate Mold Usage .................................... 245
  9.2 The Cooling System Design Process ............................. 246
    9.2.1 Calculate the Required Cooling Time ..................... 246
    9.2.2 Evaluate Required Heat Transfer Rate .................... 252
    9.2.3 Assess Coolant Flow Rate ................................ 253
    9.2.4 Assess Cooling Line Diameter ............................ 254
    9.2.5 Select Cooling Line Depth ................................ 257
    9.2.6 Select Cooling Line Pitch ................................. 260
    9.2.7 Cooling Line Routing .................................... 262
  9.3 Cooling System Designs .......................................... 266
    9.3.1 Cooling Line Networks .................................... 266
    9.3.2 Cooling Inserts ........................................... 269
    9.3.3 Conformal Cooling ........................................ 269
    9.3.4 Highly Conductive Inserts ................................ 270
    9.3.5 Cooling of Slender Cores ................................ 272
      9.3.5.1 Cooling Insert ...................................... 273
      9.3.5.2 Baffles ............................................. 274
      9.3.5.3 Bubblers ............................................ 275
      9.3.5.4 Heat Pipes ......................................... 275
      9.3.5.5 Conductive Pin ...................................... 276
      9.3.5.6 Interlocking Core with Air Channel .................. 277
    9.3.6 One-Sided Heat Flow .................................... 278
  9.4 Mold Wall Temperature Control ................................. 281
    9.4.1 Pulsed Cooling ........................................... 281
    9.4.2 Conduction Heating ....................................... 284
    9.4.3 Induction Heating ....................................... 285
    9.4.4 Managed Heat Transfer ................................... 286
  9.5 Chapter Review ................................................. 288
  9.6 References ..................................................... 289
<table>
<thead>
<tr>
<th>Chapter</th>
<th>Title</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>11.3.2</td>
<td>Ejector Blades</td>
<td>353</td>
</tr>
<tr>
<td>11.3.3</td>
<td>Ejector Sleeves</td>
<td>355</td>
</tr>
<tr>
<td>11.3.4</td>
<td>Stripper Plates</td>
<td>356</td>
</tr>
<tr>
<td>11.3.5</td>
<td>Elastic Deformation around Undercuts</td>
<td>359</td>
</tr>
<tr>
<td>11.3.6</td>
<td>Core Pulls</td>
<td>361</td>
</tr>
<tr>
<td>11.3.7</td>
<td>Slides</td>
<td>366</td>
</tr>
<tr>
<td>11.3.8</td>
<td>Early Ejector Return Systems</td>
<td>369</td>
</tr>
<tr>
<td>11.4</td>
<td>Advanced Ejection Systems</td>
<td>371</td>
</tr>
<tr>
<td>11.4.1</td>
<td>Split Cavity Molds</td>
<td>371</td>
</tr>
<tr>
<td>11.4.2</td>
<td>Collapsible Cores</td>
<td>373</td>
</tr>
<tr>
<td>11.4.3</td>
<td>Rotating Cores</td>
<td>375</td>
</tr>
<tr>
<td>11.4.4</td>
<td>Reverse Ejection</td>
<td>377</td>
</tr>
<tr>
<td>11.5</td>
<td>Chapter Review</td>
<td>378</td>
</tr>
<tr>
<td>11.6</td>
<td>References</td>
<td>380</td>
</tr>
<tr>
<td>12</td>
<td>Structural System Design</td>
<td>381</td>
</tr>
<tr>
<td>12.1</td>
<td>Objectives in Structural System Design</td>
<td>382</td>
</tr>
<tr>
<td>12.1.1</td>
<td>Minimize Stress</td>
<td>382</td>
</tr>
<tr>
<td>12.1.2</td>
<td>Minimize Mold Deflection</td>
<td>387</td>
</tr>
<tr>
<td>12.1.3</td>
<td>Minimize Mold Size</td>
<td>388</td>
</tr>
<tr>
<td>12.2</td>
<td>Analysis and Design of Plates</td>
<td>388</td>
</tr>
<tr>
<td>12.2.1</td>
<td>Plate Compression</td>
<td>389</td>
</tr>
<tr>
<td>12.2.2</td>
<td>Plate Bending</td>
<td>392</td>
</tr>
<tr>
<td>12.2.3</td>
<td>Support Pillars</td>
<td>395</td>
</tr>
<tr>
<td>12.2.4</td>
<td>Shear Stress in Side Walls</td>
<td>402</td>
</tr>
<tr>
<td>12.2.5</td>
<td>Interlocks</td>
<td>404</td>
</tr>
<tr>
<td>12.2.6</td>
<td>Stress Concentrations</td>
<td>407</td>
</tr>
<tr>
<td>12.3</td>
<td>Analysis and Design of Cores</td>
<td>410</td>
</tr>
<tr>
<td>12.3.1</td>
<td>Axial Compression</td>
<td>410</td>
</tr>
<tr>
<td>12.3.2</td>
<td>Compressive Hoop Stresses</td>
<td>412</td>
</tr>
<tr>
<td>12.3.3</td>
<td>Core Deflection</td>
<td>414</td>
</tr>
<tr>
<td>12.4</td>
<td>Fasteners</td>
<td>417</td>
</tr>
<tr>
<td>12.4.1</td>
<td>Fits</td>
<td>417</td>
</tr>
<tr>
<td>12.4.2</td>
<td>Socket Head Cap Screws</td>
<td>422</td>
</tr>
<tr>
<td>12.4.3</td>
<td>Dowels</td>
<td>424</td>
</tr>
<tr>
<td>12.5</td>
<td>Review</td>
<td>426</td>
</tr>
<tr>
<td>12.6</td>
<td>References</td>
<td>428</td>
</tr>
</tbody>
</table>
## 13 Mold Technologies

13.1 Introduction ........................................ 429
13.2 Coinjection Molds .................................. 431
   13.2.1 Coinjection Process ......................... 431
   13.2.2 Coinjection Mold Design ................. 433
13.3 Gas Assist/Water Assist Molding .............. 434
13.4 Insert Molds ....................................... 437
   13.4.1 Low Pressure Compression Molding .... 437
   13.4.2 Insert Mold with Wall Temperature Control 439
   13.4.3 Lost Core Molding ......................... 441
13.5 Injection Blow Molds ............................ 443
   13.5.1 Injection Blow Molding .................... 443
   13.5.2 Multilayer Injection Blow Molding .... 445
13.6 Multishot Molds .................................. 447
   13.6.1 Overmolding .................................. 447
   13.6.2 Core-Back Molding ......................... 449
   13.6.3 Multi-station Mold ......................... 451
13.7 In-Mold Labeling .................................. 453
   13.7.1 Statically Charged Film ................. 454
   13.7.2 Indexed Film ............................... 455
13.8 Review ........................................... 456
13.9 References ........................................ 457

## 14 Mold Commissioning

14.1 Mold Commissioning Objectives .................. 459
   14.1.1 Certify Mold Acceptability .............. 459
   14.1.2 Optimize Molding Process and Quality .... 461
   14.1.3 Develop Mold Operation and Maintenance Plans 461
14.2 Commissioning Process .......................... 462
   14.2.1 Mold Design Checklist .................... 465
   14.2.2 Component Verification .................... 465
   14.2.3 Mold Assembly ............................ 466
   14.2.4 Mold Final Test ............................ 466
   14.2.5 Preliminary Molding Recommendations .... 467
14.3 Molding Trials .................................... 470
   14.3.1 Filling Stage ............................... 471
   14.3.2 Packing Stage ............................. 473
   14.3.3 Cooling Stage ............................. 475
14.4 Production Part Approval ........................................ 476
  14.4.1 Quality Assurance ........................................ 476
  14.4.2 Gauge and Process Repeatability & Reproducibility ........ 477
  14.4.3 Image-Based Dimensional Metrology ....................... 479
  14.4.4 Process Capability Evaluation .............................. 481

14.5 Mold Maintenance ........................................... 485
  14.5.1 Pre-Molding Maintenance ................................. 487
  14.5.2 Molding Observation and Mold Map ....................... 488
  14.5.3 Post-Molding Maintenance ............................... 489
  14.5.4 Scheduled Regular Maintenance ......................... 489
  14.5.5 Mold Rebuilding ........................................ 490

14.6 Summary .................................................. 491

14.7 References ................................................ 493

Appendix ....................................................... 495

Appendix A: Plastic Material Properties ......................... 497

Appendix B: Mold Material Properties ............................ 502
  B.1 Nonferrous Metals ........................................ 502
  B.2 Common Mold Steels ........................................ 503
  B.3 Other Mold Steels .......................................... 504

Appendix C: Properties of Coolants ................................ 505

Appendix D: Statistical Labor Data ................................. 506
  D.1 United States Occupational Labor Rates ........................ 506
  D.2 International Labor Rate Comparison .......................... 506

Appendix E: Unit Conversions .................................... 508
  E.1 Length Conversions ...................................... 508
  E.2 Mass/Force Conversions ................................... 509
  E.3 Pressure Conversions ..................................... 509
  E.4 Flow Rate Conversions ................................... 509
  E.5 Viscosity Conversions ..................................... 510
  E.6 Energy Conversions ....................................... 510

Appendix F: Estimation of Melt Velocity .......................... 511

The Author .................................................. 515

Index ....................................................... 517
I would like to thank the many students, practitioners, and colleagues who have provided their ongoing support and input to the 2nd edition, including Carol Barry, Mark Berry, Maria Virginia Candal Pazos, Yasuo Ishiwata, Steve Johnston, Shmuel Kenig, Adam Kramschuster, Francis Lai, Robert Malloy, Roger Manse, Steve Orroth, Nick Schott, Steven Silvey, and Robert Stack. I’d also like to recognize Cheryl Hamilton and Mark Smith for their patience and care in this project.

Since the publication of the 1st edition, three major trends have continued with respect to plastic product and mold design:

- First, supply-chains are tightly integrated, with rapid flow of information between the product designers, molders, and mold designers. The landscape remains highly competitive, with firms differentiated by technical capability and efficiency.

- Second, advanced manufacturing is broadly recognized as a societal strategy for improving economic growth and human well-being. Of particular note is the broad interest in rapid prototyping processes (and 3D printing in particular) for supplying mold components and even low volume production of plastic parts.

- Third, the plastics industry is under increasing public pressure to minimize environmental impact. Designers of plastic products and their molds should strive to reduce, reuse, and recycle the resources that we are so fortunate to have.

The second edition has been extensively revised while reflecting on these trends. The intent has remained to provide a practical yet reasoned engineering approach. I continue to hope that Injection Mold Design Engineering is accessible and useful to all who read it. I welcome your ongoing feedback and future cooperation.

Best wishes,

David Kazmer, P.E., Ph. D.

Dandeneau Professor for Sustainable Manufacturing
Department of Plastics Engineering
University of Massachusetts Lowell
March 2016
Preface to the 1st Edition

Mold design has been more of a technical trade than an engineering process. Traditionally, practitioners have shared standard practices and learned tricks of the trade to develop sophisticated molds that often exceed customer expectations.

However, the lack of fundamental engineering analysis during mold design frequently results in molds that may fail and require extensive rework, produce moldings of inferior quality, or are less cost effective than may have been possible. Indeed, it has been estimated that on average 49 out of 50 molds require some modifications during the mold start-up process. Many times, mold designers and end-users may not know how much money was “left on the table.”

The word “engineering” in the title of this book implies a methodical and analytical approach to mold design. The engineer who understands the causality between design decisions and mold performance has the ability to make better and more informed decisions on an application by application basis. Such decision making competence is a competitive enabler by supporting the development of custom mold designs that outperform molds developed according to standard practices. The proficient engineer also avoids the cost and time needed to delegate decision to other parties, who are not necessarily more competent.

The book has been written as a teaching text, but is geared towards professionals working in a tightly integrated supply chain including product designers, mold designers, and injection molders. Compared to most handbooks, this textbook provides worked examples with rigorous analysis and detailed discussion of vital mold engineering concepts. It should be understood that this textbook purposefully investigates the prevalent and fundamental aspects of injection mold engineering.

I hope that Injection Mold Design Engineering is accessible and useful to all who read it. I welcome your feedback and partnership for future improvements.

Best wishes,

David Kazmer, P.E., Ph. D.
Lowell, Massachusetts
June 1, 2007
Injection molding is a common method for mass production and is often preferred over other processes, given its capability to economically make complex parts to tight tolerances. Before any parts can be molded, however, a suitable injection mold must be designed, manufactured, and commissioned.

The mold design directly determines the molded part quality and molding productivity. The injection mold is itself a complex system comprised of multiple components that are subjected to many cycles of temperature and stress. There are often trade-offs in mold design, with lower-cost molds sometimes resulting in lower product quality or inefficient molding processes. Engineers should strive to design injection molds that are “fit for purpose”, which means that the mold should produce parts of acceptable quality with minimal life cycle cost while taking a minimum amount of time, money, and risk to develop.

This book is directed to assist novice and expert designers of both products and molds. In this chapter, an overview of the injection molding process and various types of molds is provided so that the mold design engineer can understand the basic operation of injection molds. Next, the layout and components in three of the more common mold designs are presented. The suggested methodology for mold engineering design is then presented, which provides the structure for the remainder of this book.

### 1.1 Overview of the Injection Molding Process

Injection molding is sometimes referred to as a “net shape” manufacturing process because the molded parts emerge from the molding process in their final form with no or minimal post-processing required to further shape the product. An operating injection molding machine is depicted in Fig. 1.1. The mold is inserted and clamped between a stationary and moving platen. The mold typically is con-
nected to and moves with the machine platens, so that the molded parts are formed within a closed mold, after which the mold is opened so that the molded parts can be removed.

![Depiction of an injection molding machine and mold](image)

**Figure 1.1** Depiction of an injection molding machine and mold, adapted from [1]

The mold cavity is the “heart” of the mold where the polymer is injected and solidified to produce the molded part(s) with each molding cycle. While molding processes can differ substantially in design and operation, most injection molding processes generally include plastication, injection, packing, cooling, and ejection stages. During the plastication stage, a screw within the barrel rotates to convey plastic pellets and form a “shot” of polymer melt. The polymer melt is plasticized from solid granules or pellets through the combined effect of heat conduction from the heated barrel as well as the internal viscous heating caused by molecular deformation as the polymer is forced along the screw flights. Afterwards, during the filling stage, the plasticated shot of polymer melt is forced from the barrel of the molding machine through the nozzle and into the mold. The molten resin travels down a feed system, through one or more gates, and throughout one or more mold cavities where it forms the molded product(s).

After the mold cavity is filled with the polymer melt, the packing stage provides additional material into the mold cavity as the molten plastic melt cools and contracts. The plastic’s volumetric shrinkage varies with the material properties and application requirements, but the molding machine typically forces 1 to 10% addi-
1.1 Overview of the Injection Molding Process

Injection melt into the mold cavity during the packing stage. After the polymer melt ceases to flow, the cooling stage provides additional time for the resin in the cavity to solidify and become sufficiently rigid for ejection. Then, the molding machine actuates the moving platen and the attached moving side of the mold to provide access to the mold cavities. The mold typically contains an ejection system with moving slides and pins that are then actuated to remove the molded part(s) prior to mold closure and the start of the next molding cycle.

A chart plotting the timing of each stage of the molding process is shown in Fig. 1.2 for a molded part approximately 2 mm thick having a cycle time of 30 s. The filling time is a small part of the cycle and so is often selected to minimize the injection pressure and molded-in stresses. The packing time is of moderate duration, and is often minimized through a shot weight stability study to end with freeze-off of the polymer melt in the gate. In general, the cooling stage of the molding process dominates the cycle time since the rate of heat flow from the polymer melt to the cooler mold is limited by the low thermal diffusivity of the plastic melt. However, the plastication time may exceed the cooling time for very large shot volumes with low plastication rates. The mold reset time is also very important to minimize since it provides negligible added value to the molded product.

To minimize the molding cycle time and costs, molders strive to operate fully automatic processes with minimum mold opening and ejector strokes. The operation of fully automatic molding processes requires careful mold design, making, and commissioning. Not only must the mold operate without any hang-ups, but the quality of the molded parts must consistently meet specification.

Figure 1.2 Injection molding process timings

Figure 1.2 also shows the possible cycle timings for a more advanced mold design using additional investment in technology. Hot runner feed systems, for example, allow the use of less plastic material while also reducing injection and pack times.
Conformal cooling and highly conductive mold inserts can significantly reduce cooling times. Molds and molding processes can also be optimized to minimize mold opening, part ejection, and mold closing times. The net result of additional engineering is a reduction in the cycle time from 30 to 18 s. While some cycle time improvements are often possible just through careful engineering design, many productivity improvements require additional upfront investment in mold materials, components, or processing.

There are also many variants of the injection molding process (such as gas assist molding, water assist molding, insert molding, two shot molding, coinjection molding, injection compression molding, and others discussed later) that can be used to provide significant product differentiation or cost advantages. These more advanced processes can greatly increase the quality of the molded parts but at the same time can increase the complexity and risk of the mold design and molding processes while also limiting the number of qualified suppliers. As such, the product design and mold design should be conducted concurrently while explicitly addressing manufacturing strategy and supply chain considerations. The cost of advanced mold designs must be justified either by net cost savings or increases in the customer’s willingness to pay for advanced product designs. Cost estimation thus serves an important role in developing appropriate manufacturing strategies and mold designs.

### 1.2 Mold Functions

The injection mold is a complex system that must simultaneously meet many demands imposed by the injection molding process. The primary function of the mold is to contain the polymer melt within the mold cavity so that the mold cavity can be completely filled to form a plastic component whose shape replicates the mold cavity. A second primary function of the mold is to efficiently transfer heat from the hot polymer melt to the coolant flowing through the mold, such that injection molded products may be produced as uniformly and rapidly as possible. A third primary function of the mold is to eject the part from the mold in an efficient and consistent manner without imparting excessive stress to the moldings.

These three primary functions—contain the melt, transfer the heat, and eject the molded part(s)—also place secondary requirements on the injection mold. Figure 1.3 provides a partial hierarchy of the functions of an injection mold. For example, the function of containing the melt within the mold requires that the mold:

- *resist displacement* under the enormous forces that will tend to cause the mold to open or deflect. Excessive displacement can directly affect the dimensions of the
moldings or allow the formation of flash around the parting line of the moldings. This function is typically achieved through the use of rigid plates, support pillars, and interlocking components.

- **guide the polymer melt** from the nozzle of the molding machine to one or more cavities in the mold where the product is formed. This function is typically fulfilled through the use of a feed system and flow leaders within the cavity itself to ensure laminar and balanced flow.

**Figure 1.3** Function hierarchy for injection molds

It should be understood that Fig. 1.3 does not provide a comprehensive list of all functions of an injection mold, but just some of the essential primary and secondary functions that must be considered during the engineering design of injection molds. Even so, a skilled designer might recognize that conflicting requirements are placed on the mold design by various functions. For instance, the desire for efficient cooling may be satisfied by the use of multiple tightly spaced cooling lines that conform to the mold cavity. However, the need for part removal may require the use of multiple ejector pins at locations that conflict with the desired cooling line placement. It is up to the mold designer to consider the relative importance of the conflicting requirements and ultimately deliver a mold design that is satisfactory.

There are significant compromises and potential risks associated with mold design. In general, smaller and simpler molds may be preferred since they use less material and are easier to operate and maintain. Conversely, it is possible to under-design molds such that they may deflect under load, wear or fail prematurely, or require extended cycle times to operate. Because the potential costs of failure are
often greater than the added cost to ensure a robust design, there is a tendency to over-design with the use of conservative estimates and safety factors when in doubt. Excessive over-designing should be avoided since it can lead to large, costly, and inefficient molds.

1.3 Mold Structures

An injection mold has many structures to accomplish the functions required by the injection molding process. Since there are many different types of molds, the structure of a simple “two-plate” mold is first discussed. It is important for the mold designer to know the names and functions of the mold components, since later chapters will assume this knowledge.

The design of these components and more complex molds will be analyzed and designed in subsequent chapters.

1.3.1 External View of Mold

An isometric view of a two-plate mold is provided in Fig. 1.4. From this view, it is observed that a mold is constructed of a number of plates bolted together with socket head cap screws. These plates commonly include the top clamp plate, the cavity insert retainer plate or “A” plate, the core insert retainer plate or “B” plate, a support plate, and a rear clamp plate or ejector housing. Some mold components are referred to with multiple names. For instance, the “A” plate is sometimes referred to as the cavity insert retainer plate, since this plate retains the cavity inserts. As another example, the ejector housing is also sometimes referred to as the rear clamp plate, since it clamps to the moving platen located towards the rear of the molding machine. In some mold designs, the ejector housing is replaced with a separable rear clamp plate of uniform thickness and two parallel ejector “rails” that replace the side walls of the integral “U”-shaped ejector housing. This alternative rear clamp plate design requires more components and mold-making steps, but can provide material cost savings as well as mold design flexibility.

The mold depicted in Fig. 1.4 is referred to as a “two-plate mold” since it uses only two plates to contain the polymer melt. Mold designs may vary significantly while performing the same functions. For example, some mold designs integrate the “B” plate and the support plate into one extra-thick plate, while other mold designs may integrate the “A” plate and the top clamp plate. As previously mentioned, some mold designs may split up the ejector housing, which has a “U”-shaped profile to house the ejection mechanism and clamping slots, into a rear clamp plate and tall
rails (also known as risers). The use of an integrated ejector housing as shown in Fig. 1.4 provides for a compact mold design, while the use of separate rear clamp plate and rails provides for greater design flexibility.

![Figure 1.4 View of a closed two-plate mold](image)

To hold the mold in the injection molding machine, toe clamps are inserted in slots adjacent to the top and rear clamp plates and subsequently bolted to the stationary and moving platens of the molding machine. A locating ring, usually found at the center of the mold, closely mates with an opening in the molding machine’s stationary platen to align the inlet of the mold to the molding machine’s nozzle. The opening in the molding machine’s stationary platen can be viewed in Fig. 1.1 around the molding machine’s nozzle. The use of the locating ring is necessary for at least two reasons. First, the inlet of the melt to the mold at the mold’s sprue bushing must mate with the outlet of the melt from the nozzle of the molding machine. Second, the ejector knockout bar(s) actuated from behind the moving platen of the molding machine must mate with the ejector system of the mold. Molding machine and mold suppliers have developed standard locating ring specifications to facilitate mold-to-machine compatibility, with the most common locating ring diameter being 100 mm (4 in).

When the molding machine’s moving platen is actuated, all plates attached to the rear clamp plates will be similarly actuated and cause the mold to separate at the parting plane. When the mold is closed, guide pins and bushings are used to
closely locate the “A” and the “B” plates on separate sides of the parting plane, which is crucial to the primary mold function of containing the melt. Improper design or construction of the mold components may cause misalignment of the “A” and “B” plates, poor quality of the molded parts, and accelerated wear of the injection mold.

### 1.3.2 View of Mold during Part Ejection

Another isometric view of the mold is shown in Fig. 1.5, oriented horizontally for operation with a horizontal injection molding machine. In this depiction, the plastic melt has been injected and cooled in the mold, such that the moldings are now ready for ejection. To perform ejection, the mold is opened by at least the height of the moldings. Then, the ejector plate and associated pins are moved forward to push the moldings off the core. From this view, many of the mold components are observed, including the “B” or core insert retainer plate, two different core inserts, feed system, ejector pins, and guide pins and bushings.

![Figure 1.5 View of molding ejected from injection mold](image-url)
Figure 1.5 indicates that the plastic molding consists of two different molded parts (like a cup and a lid) attached to a feed system. This mold is called a two-plate, cold-runner, or two-cavity family mold. The term “family mold” refers to a mold in which multiple components of varying shapes and/or sizes are produced at the same time, most commonly to be used in a product assembly. The term “two-cavity” refers to the fact that the mold has two cavities to produce two moldings in each molding cycle. Such multicavity molds are used to rapidly and economically produce high quantities of molded products. Molds with eight or more cavities are common. The number of mold cavities is a critical design decision that impacts the technology, cost, size, and complexity of the mold; a cost estimation method is provided in Chapter 3 to provide design guidance.

In a multicavity mold, the cavities are placed across the parting plane to provide room between the mold cavities for the feed system, cooling lines, and other components. It is generally desired to place the mold cavities as close together as possible without sacrificing other functions such as cooling, ejection, etc. This usually results in a smaller mold that is not only less expensive, but is also easier for the molder to handle while being usable in more molding machines. The number of mold cavities in a mold can be significantly increased by not only using a larger mold, but also by using different types of molds such as a hot runner mold, three-plate mold, or stack mold as later discussed with respect to mold layout design in Chapter 4.

1.3.3 Mold Cross-Section and Function

Figure 1.6 shows the top view of the mold, along with the view that would result if the mold was physically cut along the section line A-A and viewed in the direction of the arrows. Various hatch patterns have been applied to different components to facilitate identification of the components. It is very important to understand each of these mold components and how they interact with each other and the molding process.
Consider now the stages of the molding process relative to the mold components. During the filling stage, the polymer melt flows from the nozzle of the molding machine through the orifice of the sprue bushing. The melt flows down the length of the sprue bushing and into the runners located on the parting plane. The flow then traverses across the parting plane and enters the mold cavities through small
gates. The melt flow continues until all mold cavities are completely filled. Chapters 5, 6, and 7 provide analysis and design guidelines for flow in the mold cavity, feed system, and gates. As the polymer melt fills the cavity, the displaced air must be vented from the mold. Some analysis and design guidelines are provided in Chapter 8.

After the polymer melt flows to the end of the cavity, additional material is packed into the cavity at high pressure to compensate for volumetric shrinkage of the plastic as it cools. The estimation of shrinkage and guidelines for steel-safe design are described in Chapter 9. Typically, the injection molding pressure, temperature, and timing are adjusted to achieve the desired part dimensions. The duration of the packing phase is typically controlled by the size and freeze-off of the gate between the runner and the cavity. During the packing and cooling stages, heat from the hot polymer melt is transferred to the coolant circulating in the cooling lines. The heat transfer properties of the mold components, together with the size and placement of the cooling lines, determines the rate of heat transfer and the cooling time required to solidify the plastic. At the same time, the mold components must be designed to resist deflection and stress when subjected to high melt pressures. Chapters 10 and 11 describe the analysis and design of the mold’s cooling and structural systems.

After the part has cooled, the molding machine’s moving platen is actuated and the moving half of the mold (consisting of the “B” plate, the core inserts, the support plate, the ejector housing, and related components) moves away from the stationary half (consisting of the top clamp plate, the “A” plate, the cavity inserts, and other components). Typically, the moldings stay with the moving half since they have shrunken onto the core. This shrinkage results in tensile stresses, like a rubber band stretched around a cylinder or box, that will tend to keep the moldings on the core.

After the mold opens, the ejector plate is pushed forward by the molding machine. The ejector pins are driven forward and push the moldings off the core. The moldings may then drop out of the mold or be picked up by an operator or robot. Afterwards, the ejector plate is retracted and the mold closes to receive the melt during the next molding cycle. The ejector system design is analyzed in Chapter 12.

1.4 Other Common Mold Types

A simple two-plate mold has been used to introduce the basic components and functions of an injection mold. About half of all molds closely follow this design, since the mold is simple to design and economical to produce. However, the two-plate mold has many limitations, including:
The Mold Quoting Process

The quoting process for plastic parts can be difficult for both the mold customer and supplier. Consider the view of the mold customer. The procurement specialist for the product development team sends out requests for quotes (RFQs) to several mold makers. After waiting days or weeks, the quotes come back and the customer discovers that the development time and cost of the mold may vary by a factor of three or more. In such a case, prospective mold purchasers should ask about the details of the provided quotes and check if the costs can be reduced through product redesign. To reduce uncertainty related to pricing and capability, many prospective customers maintain a list of qualified suppliers who have been found to provide satisfactory lead times, quality, and pricing across multiple projects. Long-term trust-based partnerships can provide for rapid application and mold development by avoiding the quoting process altogether and invoicing on a labor cost plus materials cost (referred to as “cost plus”) basis.

Now consider the view of the mold supplier. The mold designer may need to invest significant time developing a quote that may have a relatively small chance of being accepted. Sometimes, the mold designer may have to redesign the product and perform extensive analysis to provide the quote. While the quote may seem high to the prospective customer, the design may correspond to a mold of higher-quality materials and workmanship that can provide a higher production rate and longer working life than some other, lower-cost mold. This more expensive mold may quickly recoup its added costs during production.

From time to time, mold makers and molders will adjust their quote based on whether or not they want the business. If the supplier is extremely busy or idle, then the estimated number of hours and/or hourly rate may be adjusted to either discourage or encourage the potential customer from accepting the quote. Such adjustments should be avoided since the provided quote does not represent the true costs of the supplier, which would become the basis for future engagements between the mold supplier and the customer. Thus, the development of a long-term and mutually beneficial partnership will begin with justifiable project quotes.
The provided mold purchase contract typically states payment and delivery terms for the mold(s) and perhaps even the molded part(s). A typical mold purchase agreement may specify that the cost of a mold is paid in three installments:

- the first third: on acceptance of the quote (after which the mold base and key materials are typically purchased);
- the second third: halfway through the mold making project (often when cavity inserts have been machined); and
- the final third: upon acceptance of the quality of the molded parts.

![Figure 3.1 Schedule of mold and molding expenses](image)

After the mold is purchased, molds are typically shipped to the specified molder or the customer’s facility where the parts are molded and marginal costs are incurred on a per-part basis. The cash outlays for a typical project are plotted in Fig. 3.1 on a monthly basis. The material and processing costs in month 3 are related to molding trials by which the mold design is validated and improved; a batch of pre-production parts are sampled at this time for marketing and testing purposes. Later, monthly processing and material costs are incurred during production. Maintenance costs may appear intermittently throughout production to maintain the quality of the mold and moldings.

There has been a trend in the industry towards large vertically integrated molders with tightly integrated supply chains that can supply molded parts and even complete product assemblies. As such, the structure of the quote can vary substantially with the structure of the project and business requirements. With a vertically integrated supplier, there is typically an upfront fee for the costs associated with the development of the mold, followed by a fee for each molded part. To protect the supplier, contracts are typically developed that specify minimum production quantities with discounts and/or fees related to changes in the production schedule.
Some prospective mold customers may purposefully choose to disintegrate their supply chains in order to minimize the “leakage” of intellectual property. In this model, they may have one firm perform analysis or simulation of one component in the design, a second firm develop a mold for the same component, a third firm develop other designs and molds for other components in the design, yet other firms for molding different components, and then perform the assembly internally. Such a disintegrated supply chain can raise significant issues with respect to scheduling and product qualification.

Since the structure and magnitude of quotes will vary substantially with the supply chain strategy and supplier(s), a prospective buyer of plastic parts should solicit quotes from multiple vendors and select the quote from the supplier that provides the most preferable combination of design capability, molded part quality, and payment/delivery terms.

### 3.2 Cost Overview for Molded Parts

There are three main cost drivers for molded products:
1. the cost of the mold and its maintenance,
2. the materials cost, and
3. the processing cost.

Figure 3.2 provides a breakdown of these primary cost drivers and their underlying components. It is important to note that these costs do not include indirect costs such as facilities, administrative overhead, fringe benefits, or profits. However, such indirect costs may be accounted for through the adjustment of hourly rates or application of indirect cost rates.

![Figure 3.2 Cost drivers for injection molded products](image-url)
Even though most molded products have the same cost drivers, the proportion of costs varies widely by application. Figure 3.3 shows the cost breakdown for a commodity application (such as a cable tie with a production volume of 10 million pieces) and a specialty application (such as a custom electrical connector with a production volume of 100,000 pieces). While these two products are approximately the same weight, it is observed that the magnitude and proportion of costs are vastly different. The commodity part will tend to have lower costs due to economies of scale that allow (1) amortization of the mold cost across vast production quantities, (2) optimization of the molding process for lower molding costs, and (3) lower material costs associated with bulk purchases of resin. As Fig. 3.3 suggests, the material costs represent the majority of the total molded part cost in commodity applications whereas the mold/tooling costs can dominate for custom moldings with low production quantities.

For analysis, the total part cost of a molded product, $C_{\text{part}}$, can be estimated as

$$C_{\text{part}} = \frac{C_{\text{mold/part}} + C_{\text{material/part}} + C_{\text{process/part}}}{\text{yield}} \quad (3.1)$$

where $C_{\text{mold/part}}$ is the amortized cost of the mold and maintenance per part, $C_{\text{material/part}}$ is the material cost per part, $C_{\text{process/part}}$ is the processing cost per part, and $\text{yield}$ is the fraction of molded parts that are acceptable. Each of these terms will be subsequently estimated. To demonstrate the cost estimation method, each of these cost drivers is analyzed for the laptop bezel shown in Fig. 3.4. The example analysis assumes that 1,000,000 parts are to be molded of ABS from a single-cavity hot runner mold. Some relevant application data required to perform the cost estimation is provided in Table 3.1.
Table 3.1 Laptop Design Data

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Laptop bezel</th>
</tr>
</thead>
<tbody>
<tr>
<td>Material</td>
<td>ABS</td>
</tr>
<tr>
<td>Production quantity</td>
<td>1,000,000</td>
</tr>
<tr>
<td>$L_{\text{part}}$</td>
<td>240 mm</td>
</tr>
<tr>
<td>$W_{\text{part}}$</td>
<td>160 mm</td>
</tr>
<tr>
<td>$H_{\text{part}}$</td>
<td>10 mm</td>
</tr>
<tr>
<td>$A_{\text{part_surface}}$</td>
<td>45,700 mm$^2$</td>
</tr>
<tr>
<td>$V_{\text{part}}$</td>
<td>27,500 mm$^3$</td>
</tr>
<tr>
<td>$H_{\text{wall}}$</td>
<td>1.5 mm</td>
</tr>
</tbody>
</table>

### 3.2.1 Mold Cost per Part

The cost of the mold for a given application is estimated in Section 3.3. Given the estimate or a quote for the mold cost, $C_{\text{total\_mold}}$, the cost of the mold per part can be assessed as

$$C_{\text{mold\_part}} = \frac{C_{\text{total\_mold}}}{n_{\text{total}}} \times f_{\text{maintenance}}$$  \hspace{1cm} (3.2)

where $n_{\text{total}}$ is the total production quantity of parts to be molded, and $f_{\text{maintenance}}$ is a factor associated with maintaining the mold. Most molders perform several levels of maintenance, including:

- preventive maintenance after every molding run,
- inspections and minor repairs on an intermittent basis,
- scheduled general mold maintenance on a quarterly or semiannual basis, and
- mold rebuilding as necessary.
The need for mold maintenance and repair is related to the number of molding cycles performed, the properties of the plastic and mold materials, the processing conditions, and the quality of the mold. As a general rule, annual maintenance costs can be estimated as 10% of the mold purchase cost [1], but will vary with the design, materials, and processing conditions in application. As the resin becomes more abrasive relative to the hardness of the mold, the wear of the mold accelerates and more maintenance is required. Conversely, a well-designed, hardened mold should exhibit lower maintenance costs when used with an unfilled low-viscosity plastic. Table 3.2 provides some maintenance estimates.

### Table 3.2  Mold Maintenance Coefficient, \( f_{\text{maintenance}} \), per Million Cycles

<table>
<thead>
<tr>
<th>Mold Material</th>
<th>Unfilled, low viscosity plastic</th>
<th>High viscosity or particulate filled plastic</th>
<th>High viscosity and fiber filled plastic</th>
</tr>
</thead>
<tbody>
<tr>
<td>Soft mold material, such as aluminum or mild steel</td>
<td>4</td>
<td>16</td>
<td>64</td>
</tr>
<tr>
<td>Standard mold steel, such as P20</td>
<td>2</td>
<td>4</td>
<td>16</td>
</tr>
<tr>
<td>Hardened surface or tool steel, such as H13</td>
<td>1</td>
<td>2</td>
<td>4</td>
</tr>
</tbody>
</table>

**Example:**

Estimate the amortized cost of the mold base per molded laptop bezel.

ABS is a moderate viscosity, unfilled material. If the mold inserts are made from D2 tool steel with a hardened surface, then a mold maintenance coefficient of 2 is estimated. Given that the mold has a single cavity, one million cycles are required. The amortized cost of the mold per molded laptop bezel (including the initial purchase cost and maintenance costs) is then estimated as:

\[
C_{\text{mold/part}} = \frac{\$75,900}{1,000,000 \text{ parts}} \cdot 2 = \$0.152/\text{part}
\]

### 3.2.2  Material Cost per Part

The cost of the material per part can be estimated as:

\[
C_{\text{material/part}} = V_{\text{part}} \cdot \rho_{\text{polymer}} \cdot \kappa_{\text{polymer}} \cdot f_{\text{scrap}}
\]

where \( V_{\text{part}} \) is the volume of the molded part, \( \rho_{\text{polymer}} \) is the density of the molded polymer at room temperature, \( \kappa_{\text{polymer}} \) is the cost of the molded polymer per unit weight, and \( f_{\text{scrap}} \) is the total proportion of material consumed including startup, defects, and scrap associated with the feed system.
Table 3.3 provides estimates of the total material consumption for various types of feed systems. A cold runner is simple and low-cost but results in molded plastic that must be either discarded or recycled. Utilizing the recycled plastic as regrind reduces the waste but incurs some cost related to the labor and energy of recycling. As later described, hot runners have the potential to significantly reduce material costs but consume significant material during start-up and so are less effective in short runs.

<table>
<thead>
<tr>
<th>Type of feed design</th>
<th>Feed system waste factor, ( f_{\text{feed_waste}} )</th>
</tr>
</thead>
<tbody>
<tr>
<td>Cold runner</td>
<td>1.25</td>
</tr>
<tr>
<td>Cold runner, fully utilizing regrind</td>
<td>1.08</td>
</tr>
<tr>
<td>Hot runner with short runs</td>
<td>1.05</td>
</tr>
<tr>
<td>Hot runner with long runs</td>
<td>1.02</td>
</tr>
</tbody>
</table>

**Example:**

Estimate the cost of the plastic material per molded laptop bezel.

Since a hot runner system is used and the production quantity is one million parts, large production runs are assumed with a feed waste factor of 1.02. Using the cost and density from Appendix A, the cost of the plastic material per molded part is estimated as

\[
C_{\text{material/part}} = 27.5 \text{ cm}^3 \cdot \left( \frac{0.01 \text{ m}}{\text{cm}} \right)^3 \cdot 1044 \frac{\text{kg}}{\text{m}^3} \cdot 2.80 \frac{\$}{\text{kg}} \cdot 1.02 = 0.082/\text{part}
\]

The cost of the plastic material per part is quite low since the part has a very low thickness (1.5 mm) and low part weight (28.7 g).

### 3.2.3 Processing Cost per Part

The processing cost per part is a function of the number of mold cavities, the cycle time, \( t_{\text{cycle}} \), and the hourly rate of the machinery and labor, \( R_{\text{molding}} \):

\[
C_{\text{process/part}} = \frac{t_{\text{cycle}}}{n_{\text{cavities}}} \times \frac{R_{\text{molding}}}{3600 \text{ s/h}}
\]  

(3.4)

The cycle time is effected primarily by the thickness of the part, \( h_{\text{wall}} \), and, to a lesser extent, by the size of the part and the type of feed system. While the cycle time will be more accurately estimated during the cooling system design, a reasonable estimate is provided by
where the cycle efficiency, $f_{\text{cycle\_efficiency}}$, is a function of the type of feed system and process that is being operated according to Table 3.4. While it is desirable to operate a fully automatic molding cell with a hot runner, many molders continue to use cold runner molds operating in semiautomatic mode.

Table 3.4 Cycle efficiency coefficient

<table>
<thead>
<tr>
<th>Type of feed system and mold operation</th>
<th>Cycle efficiency factor, $f_{\text{cycle_efficiency}}$, cold runner</th>
<th>Cycle efficiency factor, $f_{\text{cycle_efficiency}}$, hot runner</th>
</tr>
</thead>
<tbody>
<tr>
<td>Semiautomatic molding with operator removal of molded parts</td>
<td>2.25</td>
<td>2.0</td>
</tr>
<tr>
<td>Semiautomatic molding with gravity drop or high speed robotic take-out</td>
<td>1.5</td>
<td>1.25</td>
</tr>
<tr>
<td>Fully automatic molding</td>
<td>1.25</td>
<td>1.0</td>
</tr>
</tbody>
</table>

The hourly rate for the molding machine is primarily a function of the clamp tonnage, which drives the size and cost of the machine. The following model was developed relating the clamp tonnage and capability to the machine hourly rate:

$$R_{\text{molding}} = \left(43.3 + 0.095 \cdot F_{\text{clamp}}\right) \cdot f_{\text{machine}}$$  (3.6)

where $F_{\text{clamp}}$ is the clamp tonnage in metric tons (mTon), and $f_{\text{machine}}$ is a factor relating to the capability of the machine and the associated labor. This equation was derived using published U.S. national hourly rate data [2] for twelve different sized molding machines ranging from 20 to 3500 metric tons; the described model has a coefficient of determination, $R^2$, equal to 0.979.

The hourly rate data is also a function of the geographic region, machine and molder costs, and other factors. To account for these variances, the machine capability factor, $f_{\text{machine}}$, is estimated according to Table 3.5. In general, molding machines with advanced capabilities and higher clamp tonnage cost more to purchase and operate, and so command a price premium. Machines with specialized capability (such as multiple injection units or very high injection pressures/velocities) are more expensive to purchase and so likewise command a price premium per hour of operation. The cost of all auxiliaries should be added to the appropriate machine coefficient. While advanced technology can increase the hourly rate of the molding process, it should provide a net savings by improving quality and reducing the processing and materials costs. Variances due to geographic locale may be accounted for by scaling the machine factor by the labor rate data provided in Appendix D relative to the U.S. cost data.
Table 3.5  Molding Machine Capability

<table>
<thead>
<tr>
<th>Type of molding machine and labor required</th>
<th>Machine factor, $f_{\text{machine}}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Old hydraulic machine (purchased before 1985) without operator or profit</td>
<td>0.8</td>
</tr>
<tr>
<td>Standard hydraulic machine or older electric machine (before 1998) operator or</td>
<td>1.0</td>
</tr>
<tr>
<td>profit</td>
<td></td>
</tr>
<tr>
<td>Modern electric machine without operator or profit</td>
<td>1.1</td>
</tr>
<tr>
<td>Molder profit</td>
<td>Add 0.1</td>
</tr>
<tr>
<td>Take-out robot and conveyor</td>
<td>Add 0.05</td>
</tr>
<tr>
<td>Hot runner temperature control</td>
<td>Add 0.05</td>
</tr>
<tr>
<td>Gas assist control</td>
<td>Add 0.1</td>
</tr>
<tr>
<td>Injection-compression control</td>
<td>Add 0.1</td>
</tr>
<tr>
<td>Dedicated operator/assembler</td>
<td>Add 0.3</td>
</tr>
<tr>
<td>Foaming or induction heating unit</td>
<td>Add 0.3</td>
</tr>
<tr>
<td>Two-shot molding machine</td>
<td>Add 0.6</td>
</tr>
<tr>
<td>Three-shot molding machine</td>
<td>Add 0.9</td>
</tr>
</tbody>
</table>

The clamp tonnage required for molding will be analyzed during the filling system design. However, the clamp tonnage can be conservatively estimated assuming an average melt pressure of 80 MPa (11,600 psi) applied to the projected area, $A_{\text{projected}}$, of the mold cavities. If the projected area is unknown, it can be estimated as the product of the part length and width. The clamp force in metric tons, $t = 9800 \text{ N}$, is then

$$F_{\text{clamp}} = 80 \cdot 10^6 [\text{Pa}] \left( \frac{A_{\text{projected}}}{n_{\text{cavities}} \cdot L_{\text{part}} \cdot W_{\text{part}} [\text{m}^2]} \right) \cdot \frac{[t]}{9800 [\text{N}]} \quad (3.7)$$

**Example:**

Estimate the processing cost per molded laptop bezel.

The analysis assumes that a hot runner system is used with a take-out robot to fully automate the molding process. The corresponding cycle efficiency factor is 1.5. The cycle time is then estimated as

$$t_{\text{cycle}} = 4 \left[ \frac{s}{\text{mm}^2} \right] (1.5[\text{mm}])^2 \cdot 1.5 = 13.5 \text{ s}$$

If a modern electric machine is used with a take-out robot/conveyor, and a hot runner controller, then, allowing for molder profit, the machine technology factor is

$$f_{\text{machine}} = 1.1 + 0.05 + 0.05 + 0.1 = 1.3$$
The clamp tonnage is estimated as

\[
F_{\text{clamp}} = 75 \cdot 10^6 \text{[Pa]} \cdot \left(1 \cdot 0.24 \text{ m} \cdot 0.16 \text{ m}^2 \right) \cdot \left(\frac{\text{mTon}}{9800 \text{[N]}}\right) = 294 \text{ mTon}
\]

It should be noted that the true required clamp tonnage is likely less than 294 metric tons since the laptop bezel has a large window in it. The analysis, however, is conservative.

The molding machine rate is then estimated as

\[
R_{\text{molding\_machine}} = (43.3 + 0.095 \cdot 294) \cdot 1.3 = \$92.60/\text{hr}
\]

The processing cost of the molded part can then be estimated by Eq. (3.4) as

\[
C_{\text{process/part}} = \frac{13.5 \text{ s/cycle}}{1 \text{ part/cycle}} \times \frac{\$92.60/\text{hr}}{3600 \text{ s/hr}} = \$0.347/\text{part}
\]

### 3.2.4 Defect Cost per Part

There are many reasons that molded parts are rejected. Some common defects include short shot, flash, contamination, improper color match, surface striations due to splay or blush, warpage and other dimensional issues, burn marks, poor gloss, and others. Since customers demand high quality levels on the molded parts they purchase, molders often internally inspect and remove any defective parts that are molded before shipment to the customer.

The cost of these defects can be incorporated into the part cost by estimating the yield. Typical yields vary from 50 to 60\% at start-up for a difficult application with many quality requirements to virtually 100\% for a fully matured commodity product. Table 3.6 provides yield estimates according to the number of molding cycles and quality requirements.

<table>
<thead>
<tr>
<th>Total number of molding cycles</th>
<th>Low quality requirements</th>
<th>High quality requirements</th>
</tr>
</thead>
<tbody>
<tr>
<td>~10,000</td>
<td>0.95</td>
<td>0.90</td>
</tr>
<tr>
<td>~100,000</td>
<td>0.98</td>
<td>0.95</td>
</tr>
<tr>
<td>~1,000,000</td>
<td>0.99</td>
<td>0.98</td>
</tr>
</tbody>
</table>
9.4 Mold Wall Temperature Control

The analyses and designs presented for mold cooling are adequate for most injection molding applications. However, there are some applications in which the use of conventional cooling designs is unacceptable. Normally, the development of a solidified skin occurs when the hot polymer melt contacts the cold mold wall [23]. In some molding applications, the solidified skin may lead to premature freeze-off of the melt in the cavity, excessive birefringence in the molded part, or inadequate levels of gloss or surface replication. In other applications, mold wall temperature fluctuations across the surface of the mold cavity may lead to a lack of dimensional control. As such, some molding applications involving lenses, air-plane cockpit canopies, optical storage media, and fiber reinforced materials may seek to improve the quality of the moldings through dynamic control of the mold wall temperature. Several different strategies are next discussed.

9.4.1 Pulsed Cooling

One approach to controlling the mold wall temperature is to use one or more sets of cooling channels to actively heat and then actively cool the mold. One such mold design is shown in Fig. 9.28; this was developed to provide tight tolerances when molding highly sensitive plastic materials or very thin walled moldings [24]. In this pulsed cooling design, a mold cavity 7 is formed by a cavity insert 10 and a core insert 9. The core insert is purposefully designed to be as thin as possible, and surrounds an internal core 12 so as to provide a channel 14 for circulation for temperature controlled fluids. The cavity insert 10 is similarly designed to mate with the cavity plate 28 and the outer insert 29 to form channels 24 and 25.

In operation, two fluids are separately temperature-controlled with a heating device 35 and a cooling device 34; two separate fluids are recommended to reduce the cost and time associated with sequentially heating and cooling a single fluid. Prior to the injection of the polymer melt, the control valves 36 and 37 will direct the heated fluid to the inlet 18 and through the mold core via channels 14 and 15 before returning via the outlet 16; a similar heating circuit is formed for the mold cavity via elements 26, 22, 25, and 27. Once the inserts 9 and 10 are at a temperature above the freezing point of the plastic melt, the plastic melt is injected into the cavity 7. The control valves can then be actuated to direct the cooling fluids from the cooling device 34 through the same channels previously used for heating.
The success of this mold design is highly dependent on minimizing the mass of the mold steel and coolant required to form and cool the walls of the mold cavity. It is clearly desirable to minimize the thickness of the mold inserts, the length of the cooling channels and lines, and the heat transfer to adjacent mold components. In this design, air gaps 20, 29, and 38 are used to reduce the amount of heat transfer and so improve the thermal efficiency and dynamic performance of the mold; insulating sheets (not numbered) are also provided adjacent the top and rear clamp plate to minimize heat transfer to the platens. Unfortunately, the size of the cavity and the structural requirements on the mold components necessitates the use of fairly large mold components that need to be heated and subsequently cooled. The dynamic thermal response is limited.
Example:

Estimate the energy required to heat the mold core and cavity inserts depicted in Fig. 9.29 for a pulsed cooling process.

For the purpose of estimating thermal energy, the core and cavity inserts can be modeled together as a block of steel with a width and length of 100 mm and a depth of 200 mm. Given a density of steel of 8000 kg/m³, the mass of the inserts estimated as 16 kg. The amount of heat, $E$, required for a temperature change, $\Delta T$, of 100°C is:

$$E = mC_p\Delta T = 16 \text{ kg} \cdot 500 \text{ J/kg°C} \cdot 100°C = 800,000 \text{ J}$$

At a cost of $0.14 per kW h, the energy cost for heating alone is on the order of $0.03 per molding cycle. This cost does not include the energy cost required for cooling, the extended molding cycle time required for heating and cooling the mold inserts, or the added cost of the mold and auxiliary systems required for implementation. For these reasons, pulsed cooling is not commonly used except in very demanding applications.

Figure 9.29 Mold design with conduction heating
9.4.2 Conduction Heating

Given the large thermal mass of the mold and the cooling system, another strategy to control the mold wall is to use conduction heaters at or near the surface of the mold. One design is shown in Fig. 9.29; this was developed to provide a smooth surface finish to one side of a foamed plastic product [25]. The mold consists of a cavity insert 12 and a core insert 10, both including a network of cooling lines 34 and 36 as per conventional mold design. A thin metallic sheet 38 conforms to the surface of the mold cavity 12, with a thin insulating layer of oxide deposited between the sheet and the cavity insert. The thin metallic sheet 38 includes an opening 40 to deliver the plastic melt from the sprue 32 to the mold cavity 14. Electrical cable attachments 46 and 48 attach the sheet 38 to low voltage, high current electric cables 50 and 52.

Just prior to mold closure, the switch 54 is closed to pass a high current through the sheet 38. In this design, a 0.2 cm thick steel plate was used with a length and width of 30 cm and 10 cm, respectively. To analyze the heating requirements, consider a typical molded part with a heat capacity of 2000 J/kg°C, a 3 mm thickness, a melt temperature of 240°C, an ejection temperature of 100°C, and a cycle time of 30 s. In this case, the heat load imposed on the mold by the ABS melt is 28 kW/m²; given that the cooling lines are placed on two sides of the mold, the cooling power is approximately 1.4 W/cm². As such, a 30 cm by 10 cm heating plate must deliver at least 420 W simply to overcome the heat transfer to the cooling lines before the temperature of the heating plate begins to increase significantly.

It is noted that conduction heaters are widely available with power densities exceeding 250 W/cm². Such a heater, if placed on the surface of a mold cavity, could increase a 0.2 cm by 30 cm by 10 cm steel plate’s surface temperature by 200°C in 6 s. Attempts have been made to incorporate higher power, thin film heaters directly into the mold surface [26]. However, such efforts to incorporate conduction heaters into molds have not been widely successful for at least three reasons. First, the large, cyclic pressure imposed on the heater(s) by the polymer melt tends to fatigue the heaters. Second, it is difficult to configure the heater(s), mold cavity, and cooling channels to provide the uniform wall temperature required to deliver aesthetic surfaces with tight dimensional controls. Third, the heaters are located between the mold cavity and the cooling channels, tend to reduce the rate of heat transfer during cooling, and so extend the cooling time.
9.4.3 Induction Heating

Induction heating is another approach to increasing the mold wall temperature prior to mold filling, and is seeing increased application for micromolding [27], gloss [28], and strength [29]. One design is shown in Figure 9.30 [30]; this was developed to injection mold reinforced thermoplastic composites with superior surface gloss and substantially no surface defects. To reduce energy consumption and heating time, only a small portion of the mold’s surface is selectively heated by high-frequency induction heating. As shown in Fig. 9.30, a conventional injection molding machine 3 delivers polymer melt to a mold consisting of a stationary mold half 4 and a movable mold half 5.

![Figure 9.30 Mold design with induction heating](image)

Prior to mold closure and filling, a high-frequency oscillator 1 drives alternating current through an inductance coil (inductor) 2 temporarily placed near the surface(s) of the mold. When a high-frequency alternating current is passed through the inductor 2, an electromagnetic field is developed around the inductor, which subsequently generates eddy currents within the metal. The resistance of the mold metal subsequently leads to internal Joule heating of the mold surface. Traces A and B in Fig. 9.30 demonstrate the increased mold surface temperature at locations A and B caused by induction heating; traces C and D show no initial effect at location C and D away from the induction heating but later increase with the heat transfer from the injected polymer melt into the mold cavity.
As with all the previously described approaches for mold wall temperature control, molders wish to elevate the surface temperature of the mold as quickly as possible. The heating power through a high-frequency induction heating is proportional to the square of the alternating frequency, the square of the current, and the square of the coil density, among other factors. As such, the inductors must be carefully designed to locally heat the mold surface in a controlled manner to avoid an undesirable temperature distribution. For example, an inductor was made from copper tube of 5 mm diameter and wound as a spiral with a pitch of 5 mm. The distance between the surface of the metal mold and the inductor was set to 1 cm. Experiments indicated that a driving frequency of 400 kHz yielded a heating power at the mold surface on the order of 1000 W/cm², which required approximately 10 s to increase the surface of the mold by 50°C.

Compared to pulsed cooling and conduction heating, induction heating provides for increased heating rates with little added mold complexity. The primary issue in implementation is the design of the inductor, and in particular the spacing of its coil windings and their relation to the mold surfaces. If the design is improper, then the heating may be limited to low power levels. Experiments [30] indicated that a heating power less than 100 W/cm² did not significantly increase the mold surface temperature and eventually caused the overload breaker to actuate. On the other hand, when the power output exceeded 10,000 W/cm², the rate of the surface temperature increase became too steep to control such that uniform heating was no longer possible; defects such as gloss irregularities, sink marks, etc. were observed with temperature differences of more than 50°C across the surface of the mold.

9.4.4 Managed Heat Transfer

Given the difficulties associated with active mold wall temperature control, a “passive” cooling design has been developed; the term “passive” is used to imply that the mold does not utilize any external power to control the mold wall temperature. The design shown in Fig. 9.31 was specifically developed to control the mold wall temperature during the molding of optical media [31]. The mold includes two halves 12 to form a mold cavity 14. Cooling lines 20 are provided per conventional design to remove the heat from the polymer melt. However, a thermal insulating member 22 is placed between the mold halves 12 and the stampers 31 and 33. The thermal insulating member 22 is made from a low thermally conductive material, preferably a high temperature polymer, such as polyimides, polyamideimides, polyamides, polysulfone, polyethersulfone, polytetrafluoroethylene, and polyetherketone. The insulating polymer is typically spin coated in an uncured form to provide a layer with a thickness on the order of 0.25 mm and subsequently heat cured. The
stamper 33 is typically fabricated from nickel, and provides the surface details for replication while also protecting and providing the insulator with a uniform, highly polished surface during molding.

During molding, the insulating layer 22 behind the stamper 33 slows the initial cooling of the resin during the molding operation. Because of this insulation, the stamper’s temperature increases and so the skin layer retains heat longer during the mold filling stage, thereby avoiding the surface irregularities created by rapid surface cooling. The temperature of the stamper:melt interface can be controlled by specification of the process conditions as well as the layers’ thicknesses and material properties; one-dimensional cooling analysis can be used to understand the physics and assist in the design optimization. In this example, it was found that the centerline temperature 51 of the disc dictates the minimum cooling time for the part to cool below the glass transition temperature of the polymer melt. The temperature 52 at the stamper:melt interface impacts the thermal stress and pit replication on the disc’s surface and is measured. The temperature 53 in the mold behind the insulator suggests that the mold acts as a heat sink and is maintained at a substantially constant temperature.

The mold designer and process engineer should intuitively understand that the addition of an insulating layer will tend to reduce the rate of heat transfer from the melt to the mold, and therefore require extended cooling times. To alleviate this issue, the cooling lines can be operated at a lower temperature to provide for higher
rates of heat transfer after the initial heating of the stamper. Accordingly, this
design strategy provides a reasonable level of mold wall temperature control
without any additional energy consumption or control systems. However, the level
of temperature control is limited compared to the other active heating designs. In
addition, this approach may be difficult to apply to complex three-dimensional
geometries.

### 9.5 Chapter Review

Cooling system design is often not leveraged in injection mold design even though
relatively little additional investment can reap significant increases in molder pro-
ductivity. The cooling system design process includes the estimation of the cooling
time, required heat transfer rate, and coolant flow rate to subsequently determine
the cooling line diameter, depth, and pitch. Once these specifications are deter-
mined, a suitable cooling line layout can be developed that provides high and uni-
form rates of heat transfer while not interfering with other mold components. The
cooling system design must also specify the flow of the coolant through the cooling
line network as well as the design of conductive inserts and other mold elements
for achieving uniform temperatures across the molded parts.

After reading this chapter, you should understand:

- The cooling system design process, and the flow of decisions needed to rationally
  engineer a cooling system;
- How to estimate the cooling time and potential errors in this estimation;
- How to estimate the required rate of heat transfer and check this value with the
  specifications of mold temperature controllers;
- How to calculate the required coolant flow rate and check this value with the
  specifications of mold temperature controllers;
- How to estimate the minimum and maximum size of a cooling line, and select a
  final cooling line diameter;
- How to estimate the depth and pitch of the cooling lines for a specific molding
  application;
- How to layout an effective cooling line design that does not interfere with other
  mold components, or redesign the mold to provide for more effective cooling;
- How to identify and remedy cooling-related issues in molding applications, such
  as sharp corners and deep cores.
- Potential approaches for controlling the mold wall temperature within a molding
cycle.
In the next chapter, the shrinkage and warpage behavior of the solidified molding is examined. Afterwards, an ejection system design process is presented. As will be made clear, the shrinkage and ejection of the molded parts are closely linked to the cooling process.

9.6 References


Injection molding is a preferred manufacturing process given its ability to quickly and efficiently make complex products to high quality. However, it is quite common for problems to be encountered during mold commissioning given the challenge of delivering stringent yet diverse key product characteristics (KPCs) while also managing significant uncertainty related to material properties and start-up processing conditions. When problems occur, it is important to assess the root cause and associated corrective remedy. Typically there issues will arise from one of four sources: 1) material properties, 2) processing conditions, 3) product design, or 4) mold design.

Multiple tuning loops are often required to develop a mold design and molding process that provide acceptable quality levels. A significant issue with mold commissioning is that the root cause(s) and potential remedies can be subject to debate. Different decision makers may strongly advocate different remedies based on their prior experiences and financial interests. Fortunately, most companies are self-interested in long-term financial stability and so will work cooperatively as partners to resolve issues and develop more strategic partnerships. While each application is governed by the specifics of the negotiated mold purchase agreement, there are some well-known customs set forth by the Society of the Plastics Industry and other industry organizations. This chapter provides an overview of some of the most important concepts with some practical guidance.

### 14.1 Mold Commissioning Objectives

#### 14.1.1 Certify Mold Acceptability

As described in Section 2.2, it is common for the mold purchase cost to not be fully paid until the mold has been found acceptable and the customer signs off on the mold acceptance. The mold designer and mold maker appreciate prompt payment
for any balance due, so the molder and end-user of the molded products should strive to certify mold acceptance within 30 days after the mold has been delivered. Longer delays can cause financial distress with the mold maker. Furthermore, very long delays can impede corrective remedies as the mold designer and mold maker will move on to other applications and may, eventually, forget or discard details related to the mold’s development such as sketches, drawings, CNC programs, patterns, and cutting tools. For this reason, molders should plan to trial received molds within a week of their arrival.

Given the potential for conflict during mold commissioning, parties in the molded product supply chain need to be reasonable with respect to mold acceptance and the implementation of corrective remedies as needed. The molder often serves as an intermediary between the mold designer/maker and the end-user of the molded parts. As such, the molder will try to balance the interests of all parties and seek the most cost- and time-effective solutions. Molders will often try to resolve molding issues first through process changes, then material changes, and finally mold design changes. Since molders routinely maintain their inventoried molds, many molders are able to quickly perform many of the changes to the mold design. However, the molder should contact the mold designer and mold maker prior to making these changes, since modification of the mold without permission can constitute acceptance of the mold by contract.

In a best case scenario, the mold will be found acceptable as shipped. In most cases where significant mold rework is required, the mold is typically shipped back to the mold maker. The cost of the mold rework can be significant and is dependent on the needed remedy as well as the expertise of the mold designer and mold maker. The owner of the mold should budget approximately 50% or more of the initial purchase price of the mold for mold rework and maintenance. Indeed, some companies employ a mold procurement strategy of purchasing multiple copies of the cheapest molds possible, then budgeting an amount for rework equal to the full purchase cost of the molds.

In a worst case scenario, the molds are not found acceptable and the cooperating parties dispute the best course of action. In some cases, the contractual obligations may not be clear or reparations cannot be made. Then, the final payment to the mold maker is never made and the molder/end-user will seek out a third party to implement corrective remedies. The original parties may simply let the matter drop or seek legal remedy regarding financial remuneration and property ownership.
14.1.2 Optimize Molding Process and Quality

Once a mold has been found acceptable, the mold commissioning process turns to optimizing the molding process and the quality of the molded products. This optimization process is typically performed by the molder with the support and approval of the end-user. The molder is motivated to maximize their profit by maximizing the yield of acceptable products while also minimizing material consumption and cycle time. Meanwhile, the end-user of the molded parts is motivated to ensure the product quality and so needs to provide strict guidance as to acceptable quality levels during mold commissioning.

Often, purchase agreements for molded products assume annual productivity gains in injection molding. The end-user should assume that the molder will attempt to continue to improve their molding processes. Accordingly, such process optimization is best conducted in early production runs, before reference process settings and quality levels are established. Minor mold design changes are often made to facilitate process optimization. The mold designer and mold maker may or may not be involved and, if so, may charge for their services on a “cost plus” basis that accounts for their time and related expenses.

14.1.3 Develop Mold Operation and Maintenance Plans

Molders will typically work with many end-users to develop mold operation and maintenance plans. During initial mold commissioning, these “plans” can be fairly rough with significant uncertainty that needs to be resolved on an application-specific basis. The reason is that each molding application has its own molding behavior with a unique set of requirements that must be fulfilled. Indeed, each mold should be considered a custom-designed machine with distinct components, operation, and maintenance requirements.

Hundreds or thousands of parts will typically be molded during the mold commissioning process, leading to valuable experience with the operation of the mold. The molder should strive to leverage these molding trials to validate and customize the acceptance and maintenance plans. Subsequently, the mold designer and mold maker are rarely involved unless replacement parts or mold rebuilding is planned on an intermittent basis. In such cases, it may be advantageous to purchase replacement parts (e.g. pins, spare cavities/cores) with the mold. Similarly, it is standard practice to order standard mold components (for example, ejector pins, cooling plugs, nozzle heaters, etc.) that may shut-down production if damaged.
14.2 Commissioning Process

Figure 14.1 provides a flow chart of the mold commissioning process, where the parties that are typically involved are shown on the left. The mold designer and mold maker are usually responsible for an internal inspection and test before they ship the mold to the molder. The mold designer and molder should work together to determine the molding process conditions such as temperature, pressures, and timings; many of these process conditions should have been estimated early in the mold specification and design. Both these parties usually work together during the initial molding trial where the mold operation is verified. Any significant defects in the mold design or workmanship are often revealed at this time, and engineering change orders (ECOs) are issued to the mold designer/maker as needed.

Figure 14.1 Mold commissioning process
Once the initial mold verification is complete, the mold designer and mold maker have fulfilled their obligations and should be paid though they are still liable for warranty costs according to the mold purchase agreement. The molder will perform a first article inspection to fully characterize the quality of the moldings. Process capability studies are often performed to optimize the molding process, perhaps with the use of scientific molding techniques and design of experiments [1]. Engineering change orders for the mold may be requested to remedy defects in the mold design or workmanship, increase the molded product quality, or otherwise improve molding productivity. The cost of these ECOs should be paid by the party responsible for the root cause:

- Mold design change due to product design change: end-user (original equipment manufacturer, OEM)
- Mold design change due to change in the mold specification: end-user or molder
- Mold design change due to defect in mold design or making: mold designer or maker

Once the mold is fully qualified with acceptable operation and molded product quality, the standard operating procedures should be recorded with a maintenance plan. Each of these foregoing steps is described in greater detail in subsequent sections.

14.2.1 Mold Design Checklist

Figure 14.2 provides a checklist for the completed mold design. At the top of the list is a set of design documents that the mold designer should provide to the molder/owner. The mold designer might begin by reviewing the mold purchase agreement and mold specification to verify that all requirements are fulfilled in the implemented mold design. The design documentation is typically specified relative to the bill of materials (BOM). Every mold component should be listed in the bill of materials along with that component’s supplier and drawing number if custom. A full set of drawings should be delivered, including completed title blocks with material, tolerances, and finishes.

The design documentation should include a mold design report or manual describing the rationale for the mold design including analysis and simulation. Layout drawings for the feed system, water lines, and ejector systems should be provided. This mold manual should also provide layout drawings of the assembled mold from every slide and views from the parting plane; these drawings can be helpful with respect to mold maintenance. The mold manual should also provide a basic process setup sheet with the estimates used for mold design. Drawings of the molded parts, both isometric and orthogonal views, should be provided with critical to quality
attributes indicated. If the mold includes a hot runner system, then the hot runner drawings and instructions should also be provided with the mold manual. All this information should be provided in native electronic CAD format unless otherwise agreed to.

### 14.2.2 Component Verification

Mold designers/makers often take pride in their work and will typically fully assemble the mold prior to inspection by the molder/owner. Molders are often tempted to immediately take the fully assembled mold and begin molding trials. However, if the mold is to be used for long-term production, then a thorough inspection of the mold components and their assembly is warranted. The component verification items identified in Fig. 14.2 can be performed at the mold maker prior to assembly, or at the molder/owner’s location after the mold has been disassembled. Each component in the mold’s bill of materials (BOM) should be verified with respect to its materials, finishes, treatments, and quantity. For complex molds, it is standard practice to number cores, cavities, ejector pins, etc. according to the mold drawings to facilitate assembly and maintenance of the mold. The core and cavity inserts should be carefully inspected with respect to finish, texture, and critical dimensions against the design drawings.

During mold assembly, the molder should verify that the mold is fully marked to their satisfaction. Each plate can be marked at its top corner with a “0” or the plate number (from 1 to the number of plates in the stack) to facilitate mold reassembly. Each mold plate should have its external edges chamfered, and eyebolt holes centered on its side(s). Each water line circuit should be labeled, with water line connectors per the molder specification. To interface with the molder’s machinery, the molder should verify the appropriateness of the mold’s locating ring, sprue bushing, and ejector rod knock-out pattern.
A
acceptable quality levels 461, 476, 481
acceptance sampling 476
actuation 466
actuation force 363
additional draft 38
aesthetic defect 111
aesthetics 29, 439
aesthetic surface 377
air channel 277
allowance 89
aluminum 261
aluminum 6061-T6 386
aluminum tooling 72
amorphous 313
amortized cost 46
angle pins 366, 371, 489
anisotropic shrinkage 301
anisotropy 314
annealing 103
anodizing 104
A plate 6, 94, 149
apparent shear rate 119
AQLs, See acceptable quality levels
artificial balancing 159
automatic de-gating 13, 197, 209
automatic molding 374
Automotive Industry Action Group (AIAG) 476
auxiliaries 50
auxiliary equipment 14
auxiliary systems 72
avoid uneven filling 111
axial compression
- of cores 410
axial mold opening direction 82
B
baffles 274
banana gate 209
barrel temperature 312
beam bending 394
bending 397
BHN, See Brinell Hardness Number
bill of materials 463, 464
blush 52
bolt strength
- ultimate stress 422
BOM, See bill of materials
bore diameter 365
boss 34, 355
boss design 34
B plate 6, 94
branched runners 188
breakeven analysis 69
Brinell Hardness Number 20, 101
bronze gib 366
bubbler 275, 373
buckling 320, 323, 350
buckling constraint 352
bulk temperature 123
burn marks 52, 125, 135, 227
business development 23

C
CAD, computer aided design 18
cam 372, 374
carbon black 314
carburizing 104
case hardening 103
cashew gate 209
cavities
- shutting off during molding 486
cavity complexity 59
cavity cost estimation 55
cavity discount factor  62

cavity filling analysis  109

cavity finishing cost  63

cavity insert  79, 80, 87

cavity insert retainer plate  6

cavity layout  91

cavity machining cost  58

cavity retainer plate  372

cavity set cost  56

chamfer  348

chamfers  35

changeover, See  switchover

change-over times  14

checklist
  - for mold design inspection  463
  - for mold layout design  106

cheek  88, 372, 402

circular layout  92

clamp force  186

clamp tonnage  50, 79, 97, 110, 130

class  72

Class 101 mold  19, 99

Class 103 mold  20

clearance  356

closed loop control  190

coefficient of friction (COF)  104

coefficient of linear thermal expansion  296

coefficient of thermal expansion  291, 314

coefficient of volumetric thermal expansion  295

coinjection  431

coinjection mold design  433

coinjection molding  431

cold runner  49, 50, 142, 144, 180, 185

collapsible cores  373

color change  14, 71, 175, 211

color matching  29

color streaking  211

common defects  52

complexity factor  58

compressibility  291, 294, 296

compression  397

compression molding  437

compression spring  368

compressive stress  340, 389, 421
  - on cores  411

computed tomography system  481

computer aided design  18

computer simulation  317, 322, 472

concurrent engineering  17

conduction heating  284

conductive inserts  270

conductive pin  276

conformal cooling  269

constraints  426

contamination  52

continuous improvement  493

contoured ejector pins  347

convective boundary  251

coolant  11

coolant flow rate  253

coolant manifolds  266

coolant temperature  311

cooling  297
  - air channel  277
  - baffle  274
  - complexity  245
  - conductive pin  276
  - cooling line depth  257
  - cooling line pitch  260
  - cooling line routing  262
  - cooling power  252
  - cooling time estimate  250
  - heat pipe  275
  - heat transfer  243
  - heat transfer coefficient  250
  - insulating layer  280
  - internal manifold  267
  - minimum time  248
  - mold-making cost  244
  - parallel setup  267
  - post-mold  292
  - reliability  245
  - required coolant flow rate  253
  - series setup  266
  - shrinkage  292
  - system design  243, 246, 266
  - temperature distribution  265
  - temperature gradient  271
  - turbulent flow  255
  - wall temperature  244

cooling circuit  266

cooling insert  269, 273

cooling line  11, 87
  - layout  91
  - maintenance  490
  - networks  266

cooling plugs  256

cooling stage  3

cooling system  49, 65, 243

cooling system cost  66

cooling time  3, 11, 173, 243, 251, 284, 311, 447, 475
  - estimate  246
copper 261
core 11, 410
- minimum wall thickness 412
- slender 414
core back 447
core-back molding 449
core bending 414
core deflection 414
core height 415
core insert 79, 80, 87
core insert retainer plate 6
core inserts
- with stripper plate 357
core pull 330, 361
- actuators 330
corner design 34
corrosion
- in cooling lines 490
cost drivers 45
cost estimates 22
cost plus 43, 461
cp, See process capability index
cpk, See process capability index
cracks 385, 408
- in molds 491
critical milestones 27
critical stress 116
Cross-WLF model 115
CTE, See coefficient of thermal expansion
Cu 940 270
cycle efficiency 50
cycle efficiency factor 50, 51
cycle time 3, 16, 27, 49, 243, 284, 446, 450
- reduction 473, 475
cycle time estimate 251
cylic stresses 385
deflection 372, 389, 394
- side walls 403
deflection temperature under load 248
degradation 97
delivery terms 44
density 246, 314
design changes 323
design for assembly 23, 30
design for injection molding 31
design for manufacturing 23, 30
design for manufacturing and assembly 291
design iterations 17
design of experiments 306, 484
design requirements 25
design standards 28
detailed design 23
development time 16
diaphragm 205
diaphragm gate 206
dieseling 227
die set for mold stack height 466
differential shrinkage 31, 243, 265, 318
dimensional adjustments 89
dimensional metrology 479
- computer tomography (CT) 480
- coordinate measurement machines 480
- optical image recognition 480
dimensions 28
direct metal laser sintering 269
discount factor 56
dispute
- during mold commissioning 460
DMLS, See direct metal laser sintering
documentation
- of mold design 463
DOE, See design of experiments
double domain 294
dowels 417, 424
DPMO, See defects per million opportunities
draft angle 38
drawings
- layout of subsystems 463
- of mold design 463
drive-interference fit 419
drops 153
dry cycle 466, 470, 487
DTUL, See deflection temperature under load
dynamic melt control 190
E
early ejector return 369
ECOs, See engineering change orders
edge gate 202
EDM, See electric discharge machining
effective area 336, 337
efficiency 50
ejection 38, 292
  – coefficient of friction 335
  – internal stresses 334
  – molding machine setup 330
  – normal force 334
  – part removal system 331
  – surface roughness 335
ejection force 334, 345
  – hoop stress 337
  – pin-to-pin variations 352
  – undercuts 359
ejection forces
  – unbalanced 360
ejection stage 330
ejection system 327, 371
  – cooling interference 332
  – cost 333
  – ejection forces 330
  – mold opening 330
  – part aesthetics 332
  – part distortion 331
  – positive return 371
  – speed 331
ejection temperature 248, 336
ejector
  – layout 345
ejector assembly 327
ejector blade 353
  – buckling 354
ejector housing 6, 95, 377
ejector knock-out rod 328, 369, 396
ejector locations 110
ejector pad 346
ejector pin 181, 230, 238, 327, 350, 396
  – clearance 238
  – contoured 39
  – stepped 352
ejector plate 11, 146, 327, 369, 377
ejector retainer plate 327, 349
ejectors
  – alignment 349
  – buckling 351
  – clearance 349
  – compressive stresses 341
  – detailing 348
  – interference 347
  – number 343
  – placement 345
  – push area 341
  – push pin 342
  – shear stress 342
  – size 343
  – sliding bearing 348
  – stripper plate 356
  – total required perimeter 342
  – ejector sleeve 346, 355
  – ejector system 65
  – design process 333
  – design strategies 343
  – ejector system cost 66
  – ejector travel 94, 95
  – elastic deformation 359
  – elastic limit 359
  – elastic modulus 383
electrical connectors 466
electric discharge machining 407
encapsulated 437
endurance limit 100
design strategies 343
design strategies 343
design strategies 343
design strategies 343
design strategies 343
design strategies 343
design strategies 343
design strategies 343
design strategies 343
endurance stress 258, 385, 505
energy efficiency 20
engineering change orders 463, 467, 484
ethylene glycol 257
excessive deflection 426
external undercuts 37

F
factor of safety 384
family mold 9, 248
fan gate 204
fasteners 417
fatal flaws 17
fatigue 100, 385, 413, 426
  – in cores 413
FDM, See fused depositon modeling
feed system 48, 141, 185
  – artificially balanced 145, 170
  – branched layout 160
  – comparison 15
  – cooling time 173
  – cost 66
  – cross-sections 176
  – custom layout 162
  – diaphragm 160
  – dynamic feed control 191
  – fill times 172
  – hybrid layout 161
  – imbalances in naturally balanced 188
  – insulated runner 185
  – layouts 159
  – maximum pressure drop 143
  – maximum volume 143
- naturally balanced 160, 170
- number of turns 175
- objectives 156
- optimization 166
- pressure drop specification 167
- primary runners 147
- radial layout 160
- residence time 175
- secondary runners 147
- self-regulating runners 192
- stack mold 186
- standard runner diameters 183
- steel safe design 184
- sub-runners 216
- tertiary runners 147
- volume 165
- waste factor 49
fidelity
- of quality measurements 479
fillers
- carbon black 314
- glass bead 314
- glass fiber 314
- mica 314
- rubber 314
fillet 35
filling 297
- complete cavity 110
filling patterns 133
filling pressure 128
filling profile
- of injection velocity 122, 467, 472
filling stage 2, 10
filling time 3, 109, 125, 469
finger 435
finishing method 36
finishing rates 63
finishing time 63
first article inspection 17, 463, 476
fit 417
fit for purpose 1
fits 417
- apparent diameter 418
- clearance 417
- insertion force 420
- interference 418
- locational-clearance 425
- locational-interference 425
- locational-transitional 425
- retention force 419
- unilateral hole basis 418
- using dowels 424
fixed core pin 355
flash 52, 83, 125, 228, 475
flash gate 205
flashing 387
flow channel 127, 436
flow leaders 135, 138, 416
flow length 213
flow rate 142
fluid assist 431
fluid assisted molding 434
foam 432
freeze-off 11
fully automatic 50, 182
fully automatic molding 13, See also injection molding
fused deposition modeling 72, 269
G
gantry robots 331
gas assist 431
gas assist molding 434
gas trap 134, 145
gas traps 229, 239
gate 11, 146, 197
- ring 416
gate freeze time 221, 223, 469, 473
gate types 216
gate well 201
gating
- automatic de-gating 197
- comparison 216
- design recommendations 218
- diaphragm gate 205
- direct sprue 200
- edge gate 202
- fan gate 204
- film gate 205
- fine-tuning 224
- flash gate 205
- freeze time 222
- gating location 213
- no-flow temperature 222
- objectives 197
- pack time 199
- pin-point gate 201
- pressure drops 219
- shear rates 198, 217
- submarine gate 209
- tab gate 203
- thermal gate 209
- thermal sprue gate 211
- tunnel gate 206
- vestige 198
gating design 197, 213
gating flexibility 12, 14, 16
gating location 109, 127
gauge repeatability and reproducibility 309, 477, 478
gauge R & R, See gauge repeatability and reproducibility
geometric complexity 58
geometric distortion 31
gibs 371
glass bead 314
glass fiber 314
glass filled 301
gloss 281, 285, 286
gloss level 29
grid layout 92
guides
– for ejector blades 354
gusset 34

H
H13 steel 102
Hagen-Poiseuille 163, 257
hardness 101
HDT 248
hear stresses 114
heat conduction 246
heat content
– of moldings 252
heat deflection or distortion temperature 248
heater resistance 466
heat flux 260
heating element 439
heat load 284
heat pipes 275
heat transfer 11
– insulating layer 287
heat transfer coefficient 250
heel block 368
height allowance 88
height dimension 87
helix 375
hesitation 111
hoop stress
– in cores 412
hot runner 14, 46, 64, 142, 153, 185, 209
– color change 144
– maintenance 489, 490
– residence time 144
– turn-over 144
hot runner mold 14, 16
hot runner system
– configurations 155
– H manifold 155
– stacked manifolds 155
– straight-bar 155
– X manifold 155
hot spots 270
hot sprue bushing 14, 153
hourly rate 49
hourly wage 62
hybrid layout 93
hydraulic actuators 364
hydraulic diameter 177

I
improper color match 52
increased molding productivity 14
indexing head 445
indirect costs 45
induction heating 285
initial investment 16
injection blow molding 443
injection blow molds 443
injection compression 373, 432, 435
injection decompression 435
injection mold 4, 6
injection molding
– cooling stage 475
– filling stage 471
– fully automatic molding 487
– packing stage 472
– process capability 481
– semiautomatic mode 487
injection molding process 1, 2, 16
injection molding process timings 3
injection pressure 97
– maximum 110, 471
injection velocity 471
injection velocity profiling 471
ink
– after mold rebuilding 491
– to check fits 466
in-mold film
– indexed 455
– statically charged 454
in-mold labeling 453
in-mold sensors 306
insert creation 87
insertion force 417
insert mold 437, 439
insert sizing guidelines 87
inspections 47
insulated runner 185
intellectual property 45
interlock 362
interlocking 415
interlocking core 277
interlocking features 85
internal corners 271
internal thread 41
internal threads 374
internal voids 33
isothermal boundary 249
isotropic 300
iterative mold development 16

J
jetting 125, 199, 468

K
Kentucky windage 322, 323
key product characteristics 459, 476
keyway 362
knit-line 145
knit-line location 439
KPCs, See key product characteristics

L
laminar flow 163
laser sintering 72
lay flat 110, 129, 133
layout design
- conflict 93
lean manufacturing 72, 267
length dimension 88
liability
- mold designer/maker 463
- molder 486
lifter 40
limit stress 100, 384, 386
limit switches 366
linear flow velocity 119
linear melt flow 204
linear melt velocity 157
linear shrinkage 300
linear velocity 112
locating dowel 357
locating pins 417
locating ring 7
locational-interference fit 419
lofted surfaces 85
lost core molding 41, 441
LPL, See processing limits
LSL, See specification limits
lubricity 104

M
machine capability factor 50
machining and wear performance 101
machining efficiency factor 61
machining factor 58
machining labor rate 58
machining rate 101
machining time 58
maintenance 44, 228
- venting 228
maintenance cost 16, 48
maintenance plan 461
managed heat transfer 286
manifold 153, 187, 443
- cooling 267
manufacturing strategies 72
manufacturing strategy
- for purchasing molds 460
marginal cost 69
material consumption 16
material cost per part 46, 48
material removal rate 59
materials cost 45
material supplier 316
material waste 49
maximum cavity pressure 110
maximum deflection 398
maximum diameter 255
maximum shear stress 402
maximum stroke 353
mechanisms 466, 489
melt flipper 160, 189
melt flow
- pressure drop 121
- velocity profile 120
melt front advancement 110
melt front velocity 468
melt pressure 109, 142
- injection limit 143
- maximum, due to endurance stress 258
melt temperature 294
metrology 476
MFI, melt flow index 115
mica 314
microfinish 36
minimum cooling line diameter 255
minimum draft angle 38
minimum wall thickness 129
multi-station mold 451
multivariate optimization 166

N
naturally balanced 154, 162
naturally balanced feed system 92
net shape manufacturing 1
Newtonian 219
Newtonian limit 116
Newtonian model 117, 163
nitriding 104
nominal dimensions 311
nominal shrinkage rate 312
nonuniform shrinkage 475
normal probability 478
nozzles 153

O
oil, for cooling 257
one-sided heat flow 278, 447
opening time 153
open loop control 191
operating cost 15
orientation 112
orifice diameter 95
original design manufacturer 30
original equipment manufacturer 30
over-filling 145
overmolding 280, 447
over-packing 111, 315, 469
over-pressure 413, 426

P
P20 steel 98, 100, 102, 383
packing 297
packing pressure 294, 469, 473
packing stage 2
  - gate freeze time 222
packing stage profiling 473
packing time 3, 199, 311, 473
pack pressure profiling 312, 323, 473
parison 444, 446
part cost 46
part dimensions 473
parting line 83, 84, 86
parting plane 9, 79, 80, 84, 93, 146, 228, 236, 404
parting surfaces 334
part interior 231
payment terms 27
peak clamp tonnage 131
physical vapor deposition 104
pilot production 24
pin length 352
pin-point gate 201
planetary gears 376
plastication 297
plastication stage 2
plastication time 3
plastic part design 21
plate bending 382, 392
plate compression 389
platen deflection 387
platens
  - bending 381
  plating 104, 491
polyjet printing 73, 269
polymer
  - amorphous 295
  - compressibility 296
  - semicrystalline 295
poor gloss 52
positive return 369
power-law 163, 167, 219
power law index 116, 119, 120
power law model 119
power law regime 116
PPAP, See production part approval process
preliminary quote 16
preloading 400
pressure difference 414
pressure drop 110, 113, 142, 163, 219, 255
  - annulus 179
  - channel flow 119
  - gates 198
  - in vents 234
  - tube flow 163
pressure test
  - of water lines and feed system 466
pressure transmission 14
pressure-volume-temperature 309
preventive maintenance 47
process capability, See injection molding
process capability index 479
  - rolled-up 483
processing conditions 124
  - robust 484
processing cost 44
processing cost per part 46, 49
processing limits 483
process optimization
  - of injection molding 461
process simulation 121, See also simulation
process window 483
process window development 483
product definition 22
product design 16, 23
product development process 21, 24
production data 27
production flexibility 72
production part approval process 476, 482
production planning 23, 27
projected area 51
projections 449
prototype mold 316
prototype molding 29
pulsed cooling 281
purchase agreement
- for injection molds 459, 463
- warranties for injection molds 463
purchase agreements
- for molded products 461
purchase cost 15
purge 175
purging 14
push area 340
push-pin 331, 342
- defect 475
PVT, pressure-volume-temperature behavior 294

Q
QC7 aluminum 383
QC10 aluminum 270
quality assurance 476
quality assurance methodology 477
quick ship 98
quoting process 43, 158

R
race-tracking 134, 135
radial flow 204
radial mold opening direction 81
rails 6, 327
rear clamp plate 6, 327, 395
recommended melt velocity 125, 127
reduced material consumption 14
reduce setup times 72
regulatory agencies 25
replacement parts 461
requests for quotes 43
required heat transfer rate 252
residence time 175
residual stress 111, 292
retainer plate 88, 366
return pins 327
reverse ejection 333, 377
rework
- cost of 460
Reynolds number 163, 255
RFQs, request for quote 43
rheology 115
rib design 33
root cause analysis 459
rotating cores 375
rubber 314
rule of thumb 251
runner 10, 142, 146, 149, See also feed system
- annulus 179
- efficiency 178
- full round 176
- half-round 176
- hydraulic diameter 177
- round-bottom 176
- shut-offs 182
- standard sizes 183
- trapezoidal 176
runner volume 165

S
safety margin 109
scientific molding 123, 463, 470
selective laser sintering 269
self-regulating valve 191
self-threading screws 34
semiautomatic 50
semiautomatic mode, See injection molding
semicrystalline 313
sensor
- cavity pressure 306
- cavity temperature 306
sensor stack 306
series layout 91, 159
setup sheet
- for molding 467
sharp corners 34
shear heating 468
shear rate 112, 115, 217
shear rates
- maximum 217
shear stress 112, 340, 372, 392, 405, 455
shear thinning 120
shims 466
short shot 52, 111, 125, 142, 199
short shot studies 471
shot size  97, 471
shot volume  97
shot weight stability studies  3
shrinkage 11, 112, 291, 292, 432
  - anisotropic  302
  - contractual obligation  317
  - in-mold  310, 475
  - isotropic  303
  - linear  292, 300
  - lower limit  314
  - negative  315
  - nonuniform  304
  - pack pressure profiling  312
  - post-mold  310
  - post-molding  475
  - processing dependence  311
  - recommendations  315
  - uncertainty  316
  - uniformity  323
  - upper limit  315
  - validation  306
  - volumetric  300
shrinkage analysis  293
shrinkage behavior  29
shrinkage data  316
shrinkage range  314
shut-offs  86
shut-off surface  230
side action  361
side wall
  - deflection  402
  - sides  372
  - bending due to shear  402
Sigmasoft  303
simulation
  - Moldex3D  122
  - mold filling  121
  - Moldflow  122
  - shrinkage  303
  - Sigmasoft  122
  - Simpoe  122
single cavity  14
single cavity mold  91
sink  33, 203
sink marks  286
sintered vent  240
slender  415
slender core  272, 278
slides  366
slideways  371
sliding cores  366
sliding fit  362, 432
SLS, selective laser sintering  269
snap beam  39
snap finger  39
S-N, stress-number fatigue curve  385, 505
Society of the Plastics Industry  19, 36
socket head cap screws  6, 417, 422
solidification temperature  336
solidified plug  209
solidified skin  281
solvent  448
specification limits  479
specific heat  246
specific volume  313
  - relation to shrinkage  298
SPI  See Society of the Plastics Industry
SPI finish  36
splay  52, 199, 468
split cavity  443, 446
split cavity design  82
split cavity mold  82, 334, 371
sprue  95, 142
sprue break  148
sprue bushing  10, 146, 149
sprue gate  200
sprue knock-out pin  147
sprue pickers  331
sprue pullers  12, 150, 180
SS420 steel  102
stack height  94, 96, 153, 187, 388
stack molds  186, 452
staged deployment  311
stagnant material  210
standards  19
start-up times  16
stationary half  11, 404
statistical process control  476
steady flow  113
steel safe  143, 218, 310, 316, 347
steel safe designs  184
stereolithography  72, 269
stop pins  327
strain  359, 383, 389
strength  100
stress  383
  - during ejection  359
stress concentrations
  - due to cooling lines  257
  - ejector holes  407
  - water lines  407
stress-strain behavior  383
stripper bolt  12, 150
stripper plate  12, 149, 356
structural and thermal performance  100
structural design 67, 381
- minimize stress 382
- mold deflection 387
- mold size 388
- safety factor 384
structural integrity 245
structural system design 381
- cost 67
structured development 21
submarine gate 209
sub-runners 216
sucker pins 150, 180, 209
supply chain 19, 23, 44, 158
support pillars 388, 395
support plate 6, 94, 327, 372, 395
surface area 59
surface area removal rate 505
surface finish 36
surface refinishing 491
surface roughness 36, 38
surface striations 52
surface texture 36, 37
surface treatments 103
switchover 486
- dynamics of velocity and pressure 474
- position 472, 475
- surge forward 475
switchover condition 472

T

tab gate 203
Tait equation 294
technical feasibility 27
temperature differences 286
temperature differential 266
temperature fluctuations 281
temperature gradient 243, 260, 265
temperature variation 262
tensile stress 336
test mold 316
thermal conductivity 243, 246
thermal contact resistance 264, 332
thermal contraction 291
thermal diffusivity 246
thermal expansion 291, 296
thermal gate 14, 209
thermal sprue gate 211
thermal strain 336
thermocouple 466
thermoplastic elastomer 280
thermoreactive diffusion 104

thickness 49
thin wall 125, 143, 281, 409
three-dimensional printing 72
three-plate 142, 148, 153, 180
three-plate mold 12, 13, 16, 185
thrust pads 154
tie bar 95
- tension 381
tie bar spacing 95
tight tolerance 29, 281, 305, 313, 323, 387, 389
tight tolerances 293
tolerance 28, 311
- stack-up 347, 348
tolerance limit 419
tolerances
- tight 293
- typical 293
tolerance specifications 29
tolerance stack-up 356
toll-gate process 21
top clamp plate 6
torpedo 209
total cost 68
TPE, thermoplastic elastomer 280
tuning loops 459
tunnel gate 206
turbulent flow 255
turret drives 452
two cavity 9
two-plate 142, 146, 153
two-plate mold 7, 11, 16
two-shot molding 280
type of gate 110
typical tolerance 29

U

ultimate stress 100, 383
undercut 39, 208, 359, 361, 449
- horizontal boss 39
- internal thread 41
- overhang 39
- side window 39
- snap finger 39
undercuts 334
undercutting 373
uniformly distributed 345
uniform wall thickness 31
unsupported spans 393
UPL, See processing limits
USL, See specification limits
Index

V
valve gate 187, 212
valve pin 179, 212
velocity to pressure switchover 474
vent channel 236
venting 227, 348
- analysis 228
- dead pockets 239
- defects 227
- design 229, 236
- dimensions 232
- ejector pins 238
- flashing 228
- locations 110, 229
- maintenance 228
- pressure drop 234
- relief 236
- thickness
  - maximum 235
  - minimum 233
vertically integrated molders 44
viscosity 112, 115
- Arrhenius temperature dependence 116
- Cross-WLF model 115
- Newtonian model 118
- Newtonian plateau 117
- no-flow temperature 222
- power law model 119
- power law regime 116
- WLF temperature dependence 116
viscous flow 112
volumetric flow rate 118, 120, 171
volumetric removal rate 505
volumetric shrinkage 33, 297, 473
von Mises stress 382, 407
V/P, See switchover condition

W
wall thickness 32, 109
- minimum 127
warpage 52, 111, 243, 291, 317
- avoidance strategies 323
- differential shrinkage 318
- Kentucky windage 322
- out of plane deflection 318
- pressure gradient 319
- radius of curvature 318
- sources 318
- temperature gradient 318
water assist 431
water assist molding 434
water lines 487
- maintenance 489
wear
- maintenance of 490
wear plates 372
weld line 134
width dimension 88
windage 322
window 86
witness line 83, 86, 333, 358, 375
witness mark 198, 332, 377, 446
worst case scenario 384, 423

Y
yield 46, 53, 481
yield estimates 52
yield stress 100, 383

Z
zero shear viscosity 116, 118