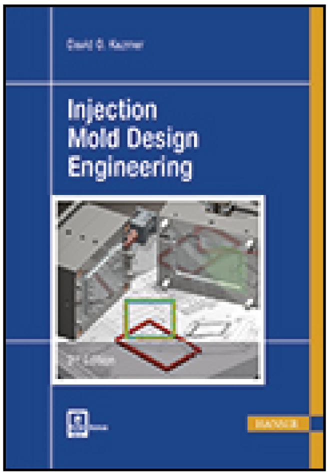
HANSER



Sample Pages

David O. Kazmer

Injection Mold Design Engineering

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

eBook ISBN: 978-1-56990-571-5

For further information and order see

www.hanserpublications.com (in the Americas)

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

Contents

6 Feed System Design 141

12 Structural System Design 381

Preface to the 2nd Edition

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

1 Introduction

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 connected 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.

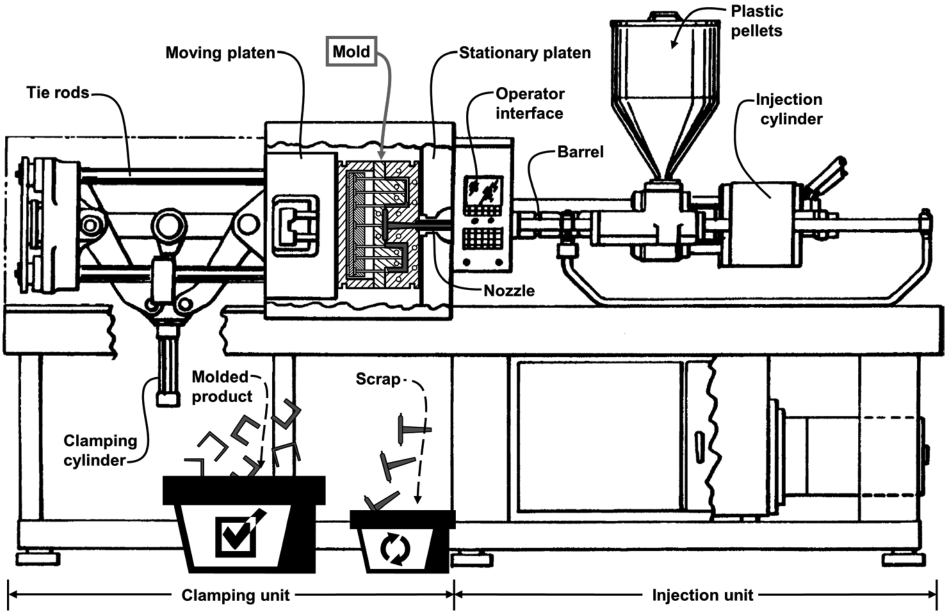


Figure 1.1 Depiction of an injection molding machine and mold, adapted from [1]

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 % additional 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.

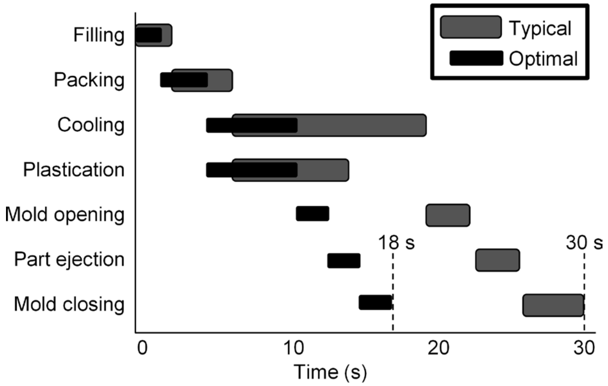


Figure 1.2 Injection molding process timings

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.

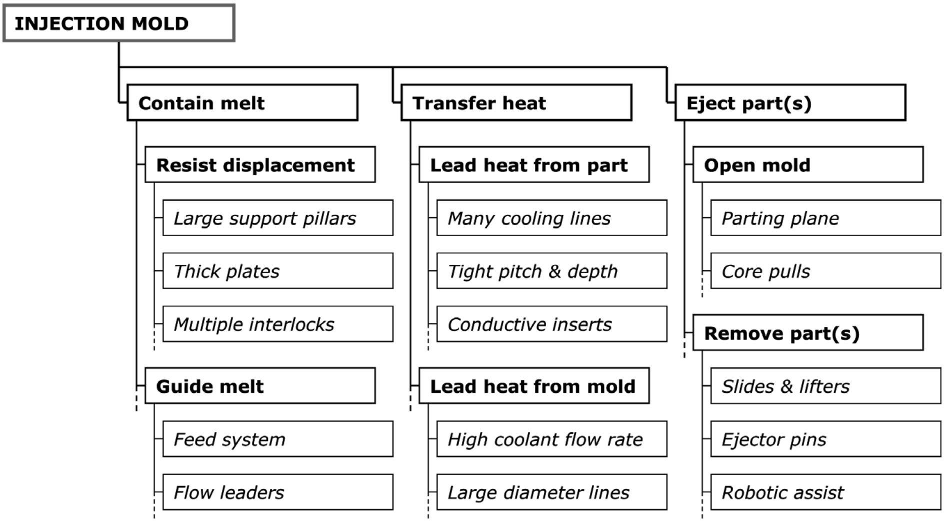


Figure 1.3 Function hierarchy for injection molds

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 underdesign 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.

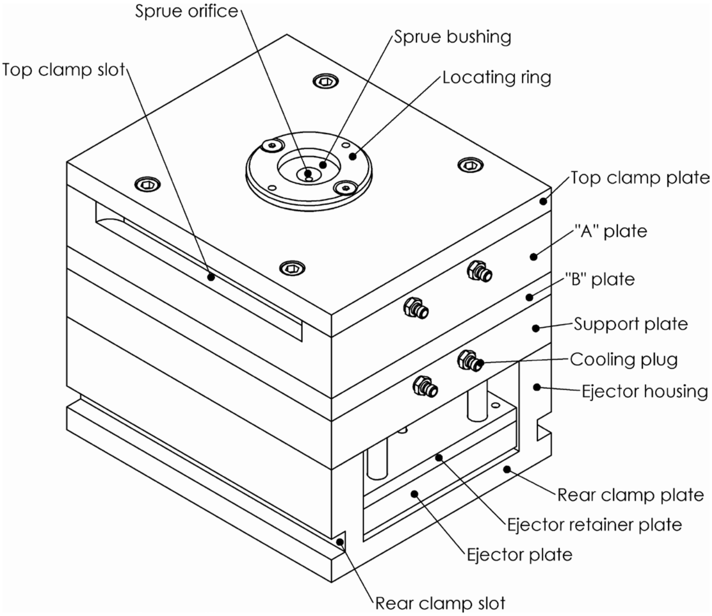


Figure 1.4 View of a closed two-plate mold

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.

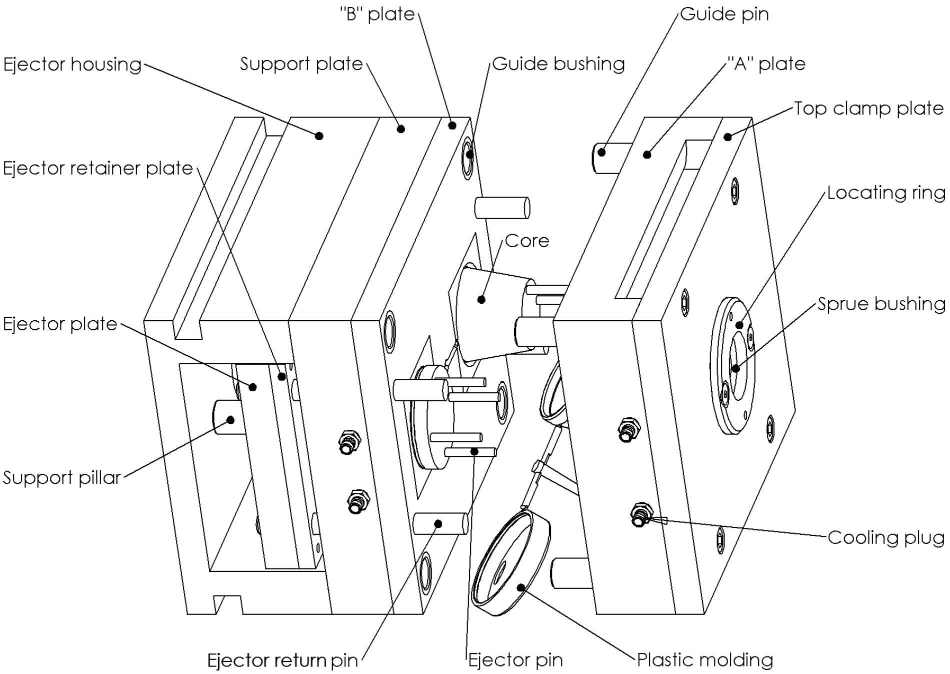


Figure 1.5 View of molding ejected from injection mold

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.

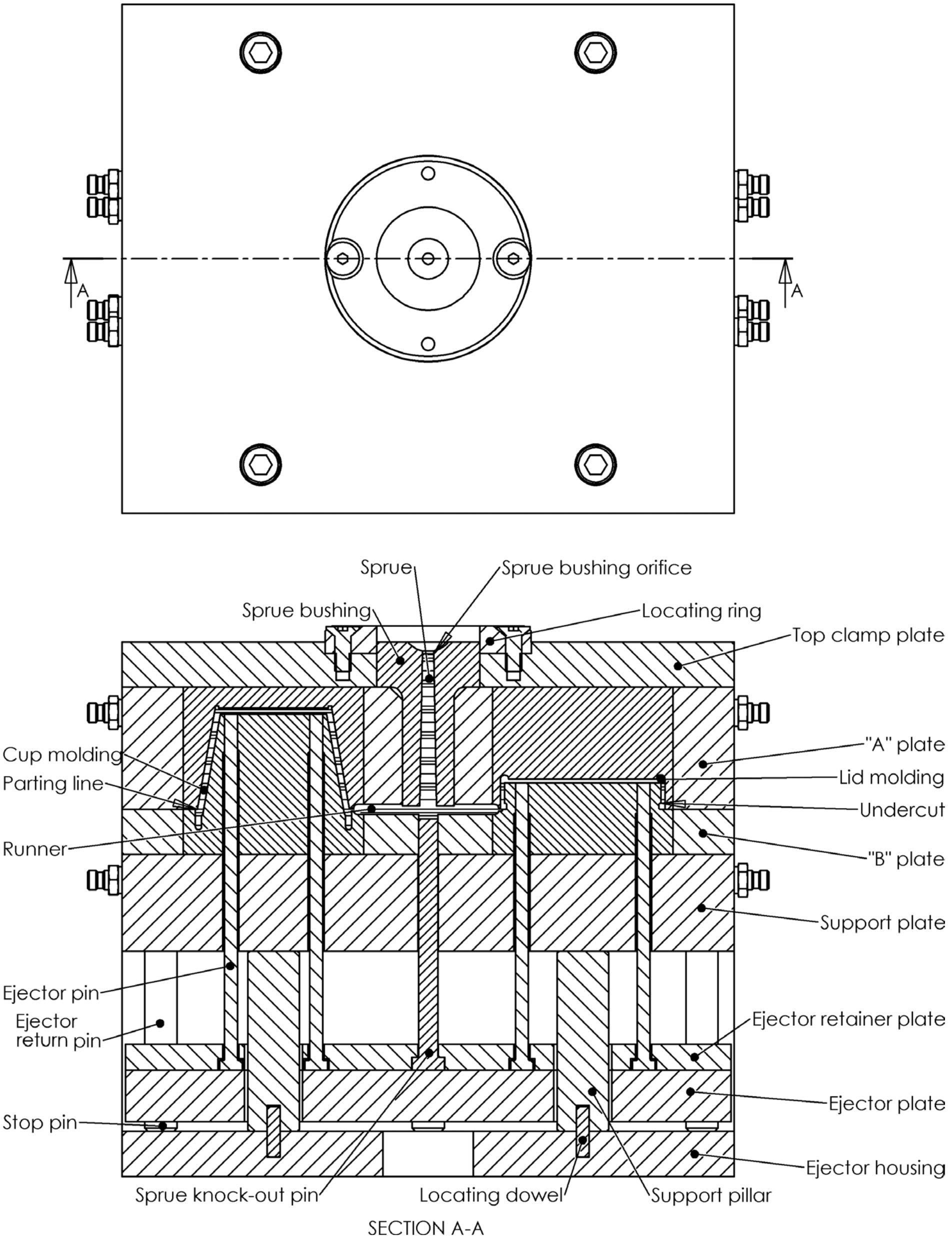


Figure 1.6 Top and cross-section views of a two-plate mold

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:

3

Mold Cost Estimation

3.1 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.

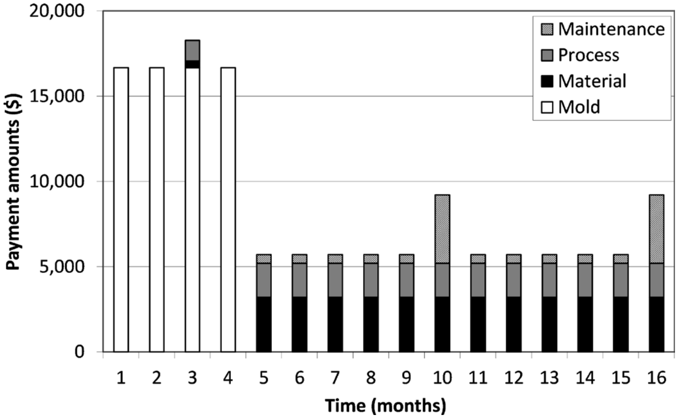


Figure 3.1 Schedule of mold and molding expenses

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.

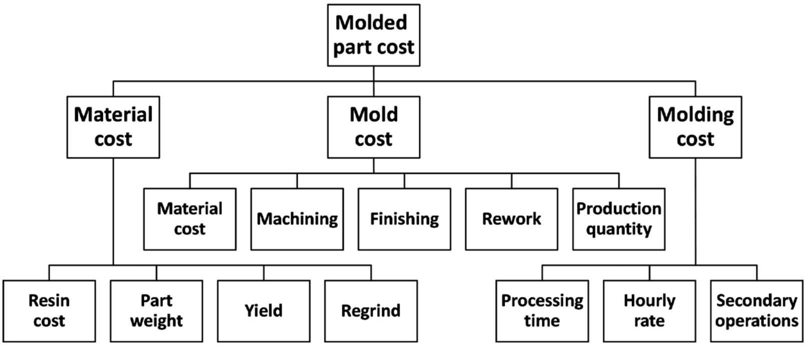


Figure 3.2 Cost drivers for injection molded products

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.

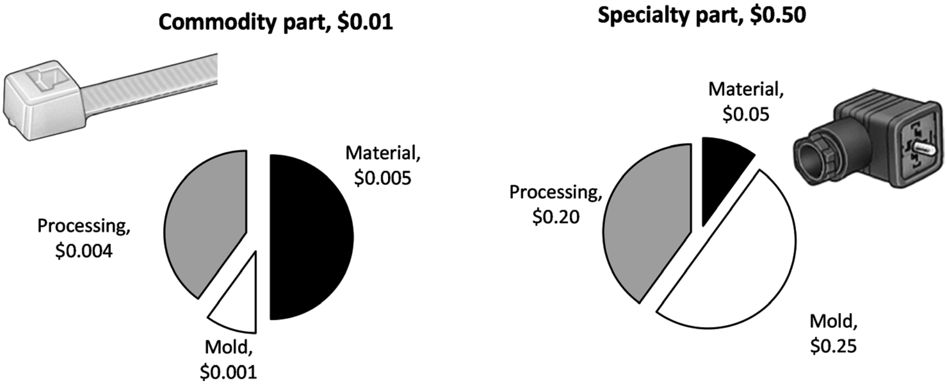


Figure 3.3 Cost comparisons for a commodity and specialty part

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

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

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 *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.

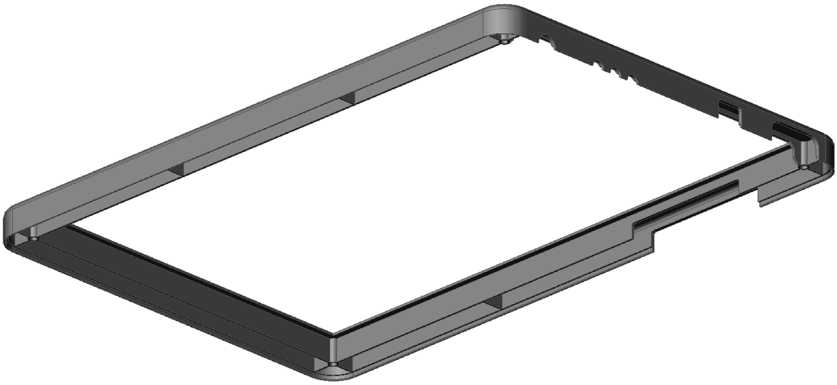


Figure 3.4 Isometric view of laptop bezel

Table 3.1 Laptop Design Data

Parameter	Laptop bezel
Material	ABS
Production quantity	1,000,000
L_{part}	240 mm
W_{part}	160 mm
H_{part}	10 mm
$A_{part\_surface}$	45,700 mm^2
V_{part}	27,500 mm^3
H_{wall}	1.5 mm

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_{total_mold}, the cost of the mold per part can be assessed as

$$C_{mold/part} = \frac{C_{total\_mold}}{n_{total}} \times f_{maintenance} \tag{3.2}$$

where n_{total} is the total production quantity of parts to be molded, and $f_{maintenance}$ 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

	Unfilled, low viscosity plastic	High viscosity or particulate filled plastic	High viscosity and fiber filled plastic
Soft mold material, such as aluminum or mild steel	4	16	64
Standard mold steel, such as P20	2	4	16
Hardened surface or tool steel, such as H13	1	2	4

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:

$$C_{\text{material/part}} = V_{\text{part}} \cdot \rho_{\text{polymer}} \cdot \kappa_{\text{polymer}} \cdot f_{\text{scrap}} \tag{3.3}$$

where V_{part} is the volume of the molded part, ρ_{polymer} is the density of the molded polymer at room temperature, κ_{polymer} is the cost of the molded polymer per unit weight, and f_{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 3.3 Material Waste Coefficient

Type of feed design	Feed system waste factor, $f_{\text{feed\_waste}}$
Cold runner	1.25
Cold runner, fully utilizing regrind	1.08
Hot runner with short runs	1.05
Hot runner with long runs	1.02

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_{cycle}, and the hourly rate of the machinery and labor, R_{molding}:

$$C_{\text{process/part}} = \frac{t_{\text{cycle}}}{n_{\text{cavities}}} \times \frac{R_{\text{molding}}}{3600\,\text{s/h}} \tag{3.4}$$

The cycle time is effected primarily by the thickness of the part, h_{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

$$t_{\text{cycle}} = 4\left[\frac{s}{\text{mm}^2}\right]\left(h_{\text{wall}}\left[\text{mm}\right]\right)^2 \times f_{\text{cycle\_efficiency}} \tag{3.5}$$

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

Type of feed system and mold operation	Cycle efficiency factor, $f_{\text{cycle\_efficiency}}$, cold runner	Cycle efficiency factor, $f_{\text{cycle\_efficiency}}$, hot runner
Semiautomatic molding with operator removal of molded parts	2.25	2.0
Semiautomatic molding with gravity drop or high speed robotic take-out	1.5	1.25
Fully automatic molding	1.25	1.0

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:

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

where F_{clamp} is the clamp tonnage in metric tons (mTon), and f_{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_{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 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

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

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, $t = 9800$ N, is then

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

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

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

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

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

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

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

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 3.6 Yield Estimates

Total number of molding cycles	Low quality requirements	High quality requirements
~10,000	0.95	0.90
~100,000	0.98	0.95
~1,000,000	0.99	0.98

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, airplane 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.

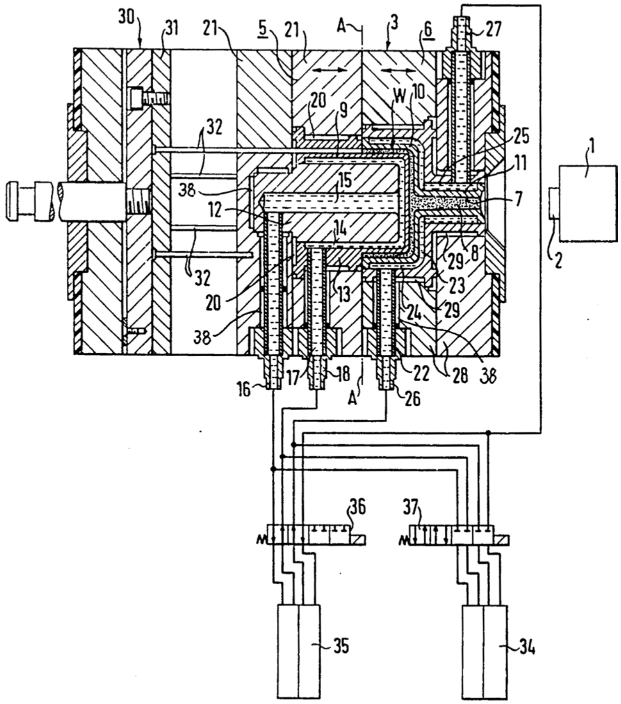


Figure 9.28 Mold design for pulsed cooling

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/m^3, the mass of the inserts estimated as 16 kg. The amount of heat, E, required for a temperature change, ΔT, of 100°C is:

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

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.

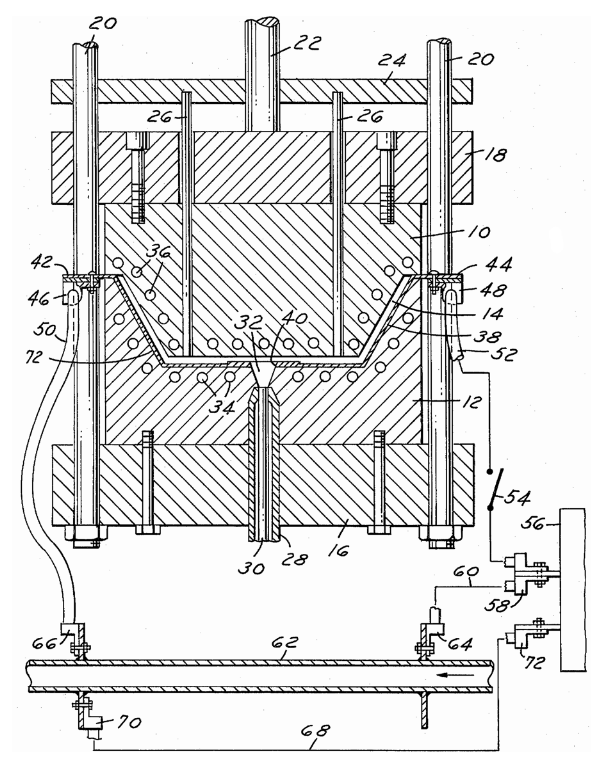


Figure 9.29 Mold design with conduction heating

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/m^2; given that the cooling lines are placed on two sides of the mold, the cooling power is approximately 1.4 W/cm^2. 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/cm^2. 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.

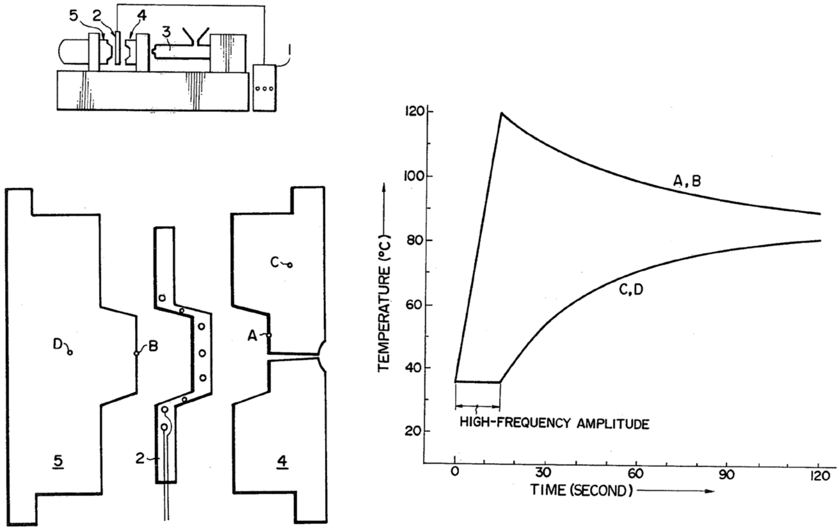


Figure 9.30 Mold design with induction heating

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/cm^2, 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/cm^2 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/cm^2, 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.

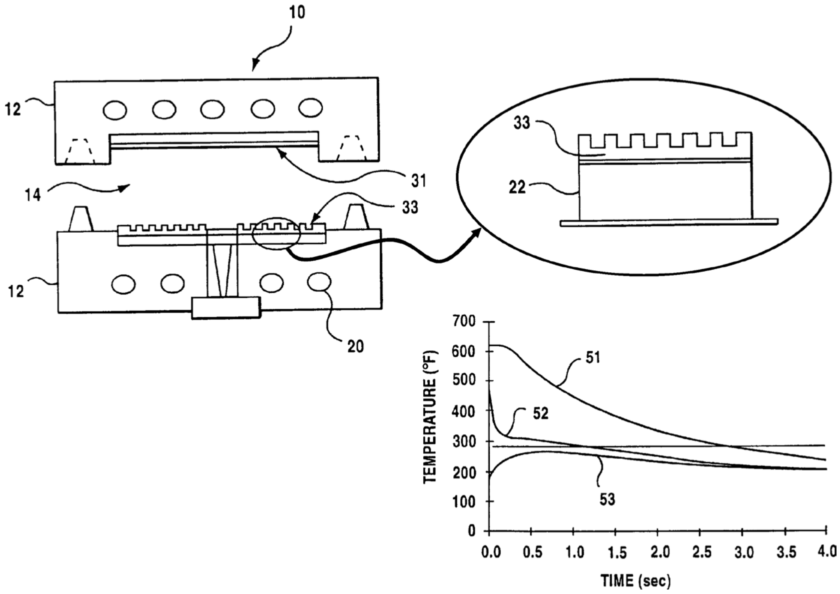


Figure 9.31 Mold design with managed heat transfer

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 productivity. 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 determined, a suitable cooling line layout can be developed that provides high and uniform 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

[1] Kramschuster, A., et al., *Quantitative study of shrinkage and warpage behavior for microcellular and conventional injection molding,* Polym. Eng. Sci., (2005) 45(10): pp. 1408–1418

[2] Kurt, M., et al., *Influence of molding conditions on the shrinkage and roundness of injection molded parts,* Int. J. Adv. Des. Manuf. Technol. (2010) 46(5–8): pp. 571–578

[3] Hatch, D., D. Kazmer, and B. Fan, *Dynamic Cooling Design for Injection Molding,* in *SPE ANTEC Tech. Papers* (2001) 47: pp. 428–432

[4] Kreith, F. and W.Z. Black, *Chapter 3: Transient Conduction, in Basic Heat Transfer,* Harper & Row (1980) pp. 136–189

[5] Xu, H. and D.O. Kazmer, *A Stiffness Criterion for Cooling Time Estimation,* Int. Polym. Process. (1999) 13(3): pp. 249–255

[6] Yu, C.J. and J.E. Sunderland, *Determination of ejection temperature and cooling time in injection molding,* Polym. Eng. Sci. (1992) 32(3): p. 191

[7] Yu, C., J. Sunderland, and C. Poli, *Thermal contact resistance in injection molding,* Polym. Eng. Sci. (1990) 30(24): pp. 1599–1606

[8] Costa, F., G. Tosello, and B. Whiteside. *Best practice strategies for validation of micro moulding process simulation,* in *Polym. Process Eng.* ‘09 Conf. (2009)

[9] Moody, L.F., *Friction Factors for Pipe Flow,* ASME Transactions (1944) 66(8): p. 671

[10] Pilkey, W.D. and D.F. Pilkey, *Peterson’s stress concentration factors,* John Wiley & Sons (2008)

[11] Timoshenko, S. and J. Goodier, *Theory of elasticity, Engineering Societies Monographs,* International Student edition, New York (1970)

[12] Menges, G., W. Michaeli, and P. Mohren, *How to Make Injection Molds,* Hanser, Munich (2001) p. 409

[13] Mikić, B., *Thermal contact conductance; theoretical considerations,* Int. J. Heat Mass Transfer (1974) 17(2): pp. 205–214

[14] Chun Wu, Y., *Lean manufacturing: a perspective of lean suppliers,* Int. J. Oper. Prod. Manage. (2003) 23(11): pp. 1349–1376

[15] Melton, T., *The benefits of lean manufacturing: what lean thinking has to offer the process industries,* Chem. Eng. Res. Des. (2005) 83(6): pp. 662–673

[16] Monden, Y., *Toyota production system: an integrated approach to just-in-time,* CRC Press (2011)

[17] Oroszlany, Á., P. Nagy, and J.G. Kovács, *Injection molding of degradable interference screws into polymeric mold,* in *Materials Science Forum,* Trans Tech Publ. (2010)

[18] Cedorge, T. and J. Colton, *Draft angle and surface roughness effects on stereolithography molds,* Polym. Eng. Sci. (2000) 40(7): pp. 1581–1588

[19] Lee, C., et al., *Rapid investment casting: direct and indirect approaches via fused deposition modelling,* Int. J. Adv. Manuf. Technol. (2004) 23(1–2): pp. 93–101

[20] Kruth, J.-P., et al., *Lasers and materials in selective laser sintering,* Assembly Automation (2003) 23(4): pp. 357–371

14 Mold Commissioning

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.

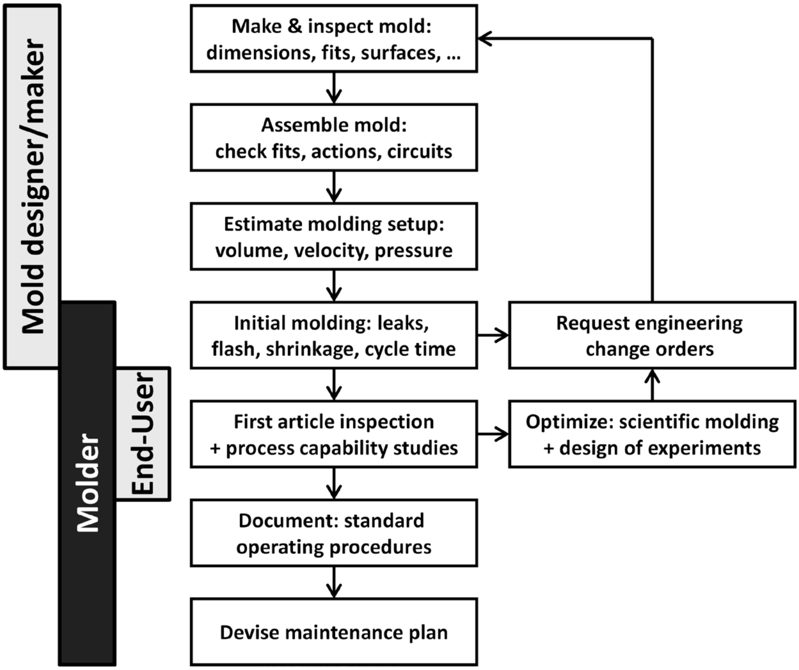


Figure 14.1 Mold commissioning process

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.

Index