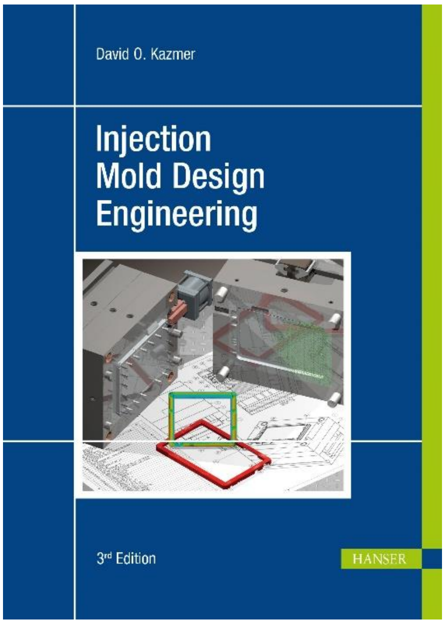
HANSER



Sample Pages

Injection Mold Design Engineering

David O. Kazmer

ISBN (Book): 978-1-56990-891-4
ISBN (E-Book): 978-1-56990-892-1

For further information and order see

www.hanserpublications.com (in the Americas)

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

Preface

This 3rd edition of *Injection Mold Design Engineering* reflects changes to address the growing adoption of additive manufacturing and information technologies in order to increase the flow and value of molded products. My intent is to provide a practical yet reasoned approach to the implementation of these new technologies.

This edition seeks to provide guidance on when and how to use additive manufacturing relative to traditional mold design, mold making, and injection molding. While some plastic part designs will transition to additive manufacturing processes for production purposes, injection molding is much more environmentally sustainable as evidenced by much lower materials, processing, energy, and labor costs. As such, a core strategy is the use of additive manufacturing for production of mold inserts to reduce cost and improve performance of production injection molds. Additive manufacturing already enables conformal cooling but we are likely to see even broader adoption with some of the new designs and supporting guidance provided in this edition.

In this post-Covid era, the increasing interest in information technology and the "metaverse" is indicative of the need for engineered mold designs that (i) are developed in a structured manner with a minimum of physical exchanges and engineering revisions, and (ii) provide high-fidelity information that supports the mold design engineering and ongoing mold use with single part tracking of each molding cycle. For these reasons, this edition addresses the use of 3D injection molding simulations throughout the mold design process. New content is also provided about the selection and use of in-mold instrumentation and synthetic data with Industry 4.0. There is little doubt that the use of such instrumentation will provide significant value and provide a means for improved molding performance and competitive differentiation.

I wish you all well and thank you for all your support across the years. Most importantly, I want to thank my wife Nancy and daughters Laura, Julia, Elizabeth, and Catherine for all their patience and love. I am amazed and grateful for all of life's wonders.

Sincerely,

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

Professor and Past Chair, Department of Plastics Engineering
University of Massachusetts Lowell
July 2022

Contents

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. Injection molding is also one of the most efficient manufacturing processes due to fast cycle times associated with high rates of cooling, low material waste, and the potential for using recycled and sustainable materials [1].

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 thermal and stress cycles. 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 minimizing the 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. A simplified injection molding machine is depicted in Figure 1.1. The mold is inserted and clamped between a stationary and a moving platen. The mold 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.

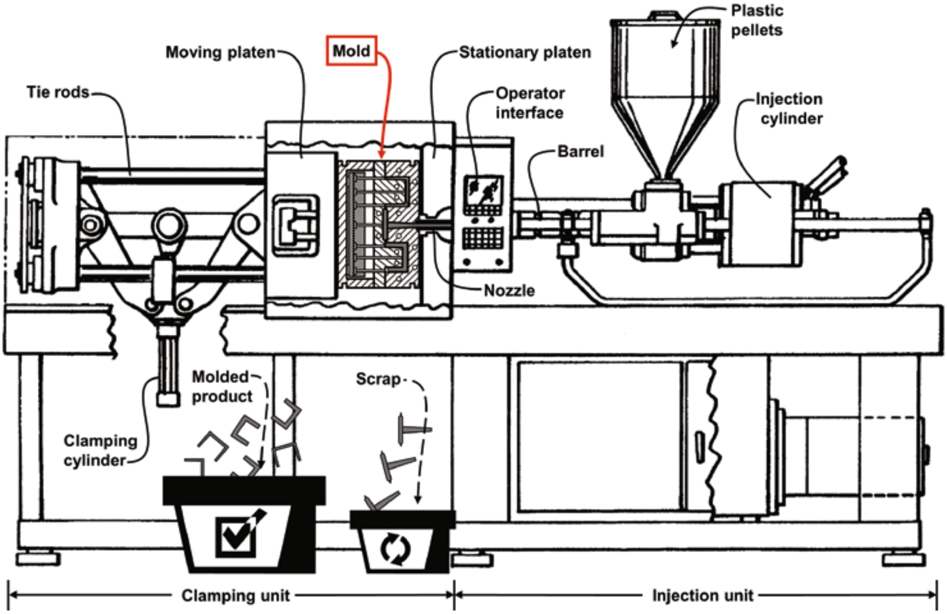


Figure 1.1 Depiction of an injection molding machine and mold, adapted from [2]

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, flake pellets, or powder 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 Figure 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.

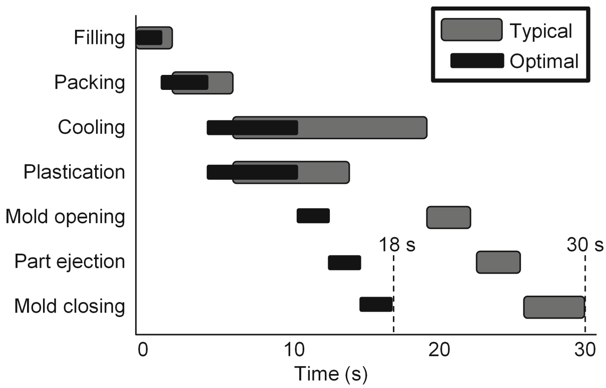


Figure 1.2 Injection molding process timings

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. Process automation including robotics can assist in further reducing the cycle time by precise synchronization of material handling equipment with the movement of the machine platen while also supporting faster takeout than traditional gravity drops. However, the operation of fully automated 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. Multivariate sensing and quality assurance is increasingly common to provide the required process and quality control.

Figure 1.2 shows likely productivity gains using a more advanced mold design with some 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. Expertise, judgement, and strategy play a significant role in each mold design application.

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 value 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 [3]. 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.

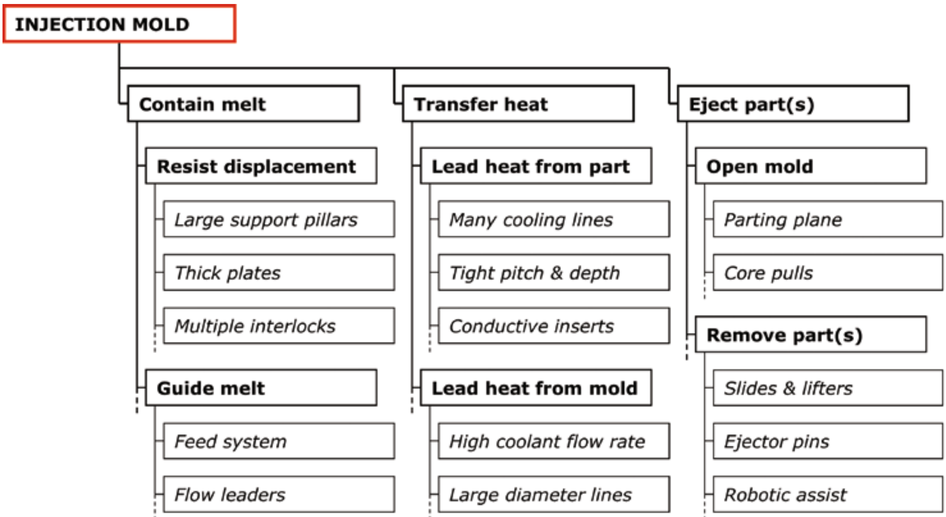


Figure 1.3 Function hierarchy for injection molds

It should be understood that Figure 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. Basic and more complex mold structures 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 Figure 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.

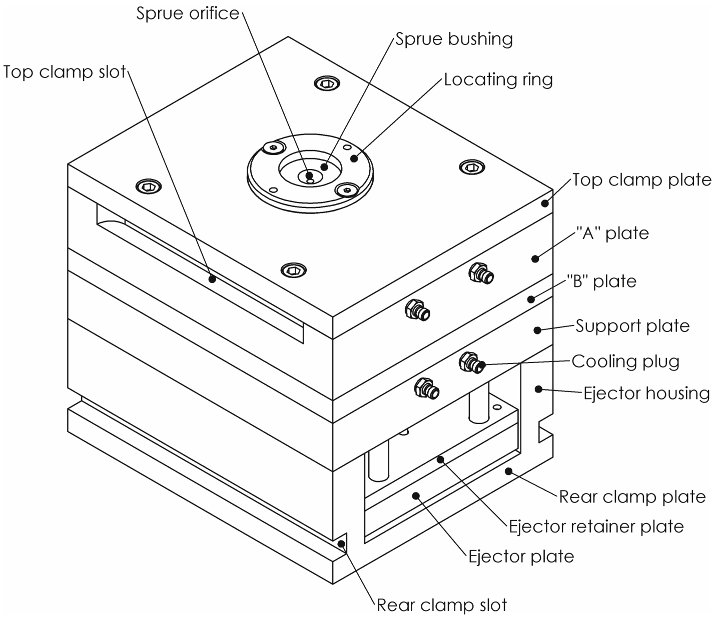


Figure 1.4 View of a closed two-plate mold

The mold depicted in Figure 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 Figure 1.4 provides for a compact mold design, while the use of separate rear clamp plate and rails provides for greater design flexibility.

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 Figure 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 between the "A" and "B" plates. 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 Figure 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 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, 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 use of the family mold ensures that the material comprising the molded components is the same, which can be important with respect to color, strength, size, and other properties. 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.

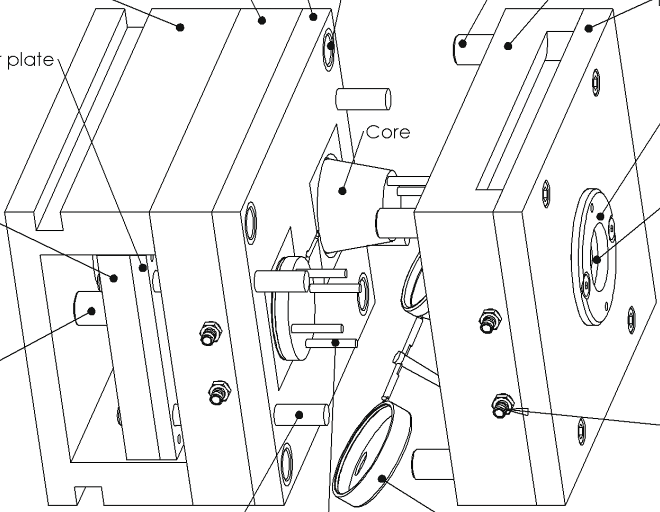


Figure 1.5 View of cup and lid moldings ejected from injection mold

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. Tight spacing of the cavities usually results in a smaller mold that is not only less expensive but 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 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 important to understand each of these mold components and how they interact with each other and the molding process.

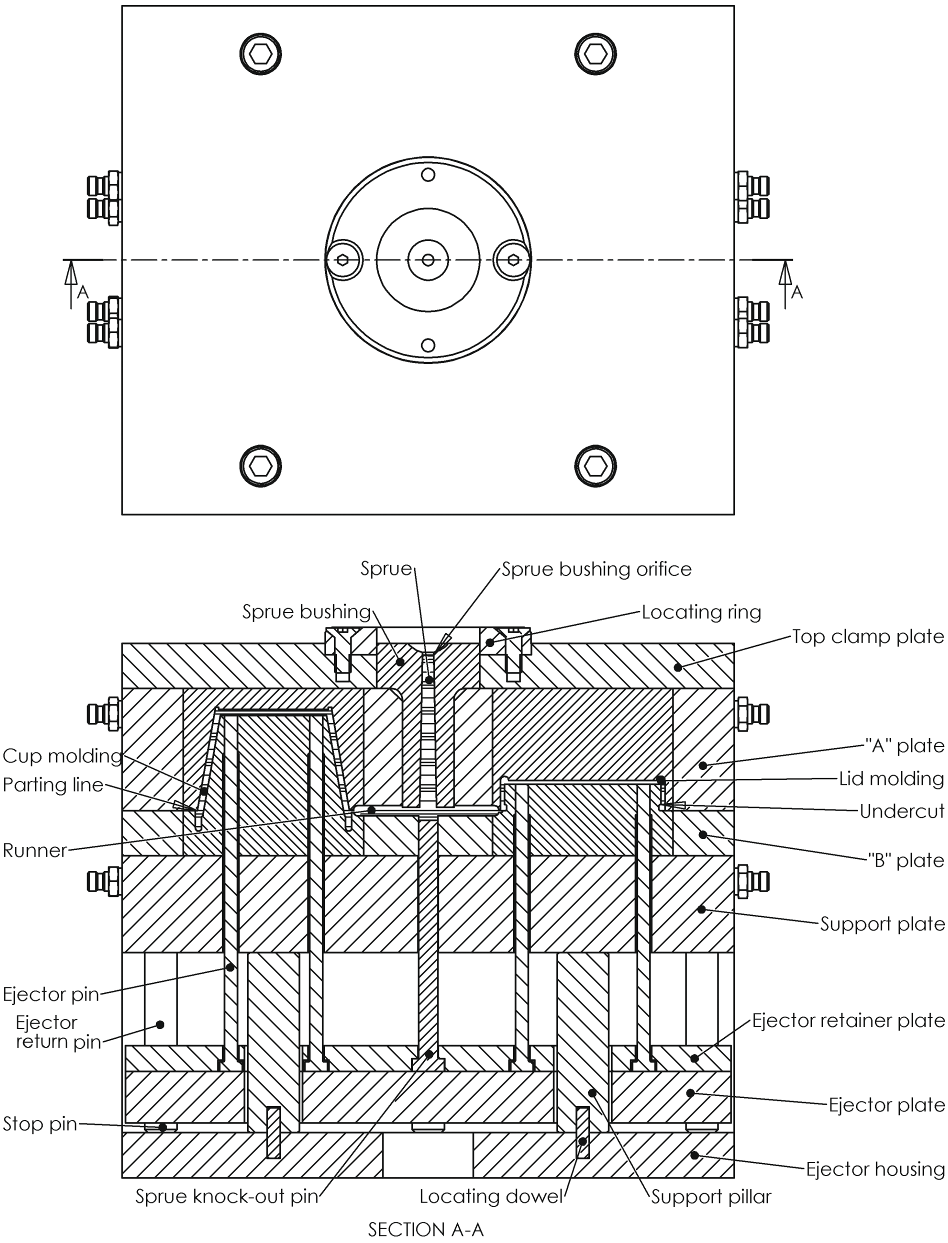


Figure 1.6 Top and cross-section views of a two-plate mold

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, determine 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 respectively 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 residual 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 it is simple to carry out and economical to produce. However, the two-plate mold has many limitations, including:

- restriction of the feed system route to the parting plane;
- limited gating options from the feed system into the mold cavity or cavities;
- restriction on the tight spacing of cavities;
- additional clamping forces imposed on the mold by the melt flowing through (and being pressurized within) the feed system;
- increased material waste incurred by the solidification of the melt in the feed system; and
- increased cycle time related to the plastication and cooling of the melt in the feed system.

For these reasons, molding applications requiring high production quantities often do not use two-plate mold designs but instead rely on more complex designs that provide lower-cost production of the molded parts. Such designs include three-plate molds, hot runner molds, stack molds, and others. Three-plate molds and hot runner molds are the next most common types of injection molds, and so are introduced next.

1.4.1 Three-Plate, Multicavity Family Mold

The three-plate mold is so named since it provides a third plate that floats between the mold cavities and the top clamp plate. Figure 1.7 shows a cut section of a three-plate mold that is fully open with the moldings still on the core inserts. As shown in Figure 1.7, the addition of the third plate provides a second parting plane between the “A” plate assembly and the top clamp plate for the provision of a feed system that traverses parallel to the parting plane. During molding, the plastic melt flows out the nozzle of the molding machine, down the sprue bushing, across the primary runners, down the sprues, through the gates, and into the mold cavities. The feed system then freezes in place with the moldings.

When the mold is opened, the molded cold runner will stay on the stripper plate due to the inclusion of sprue pullers that protrude into the primary runner. As the mold continues to open, the stripper bolt connected to the “B” plate assembly will pull the “A” plate assembly away from the top clamp plate. Another set of stripper bolts will then pull the stripper plate away from the top clamp plate, stripping the molded cold runner off the sprue pullers. The ejector plate may be designed and actuated as in a traditional two-plate mold to force the moldings off the core.

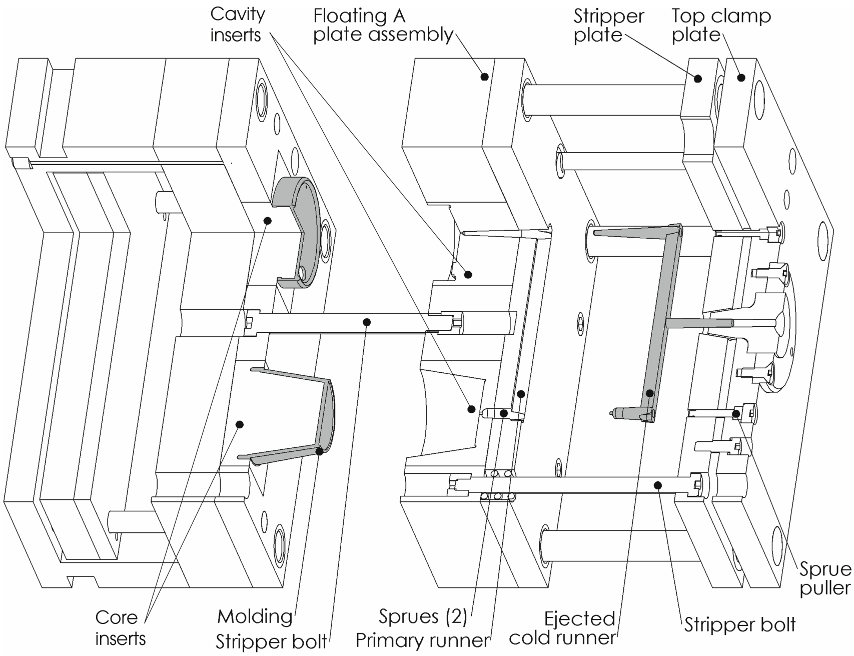


Figure 1.7 Section of an open three-plate mold

The three-plate mold eliminates two significant limitations of two-plate molds. First, the three-plate mold allows for primary and secondary runners to be located in a plane above the mold cavities so that the plastic melt in the cavities can be gated at any location. Such gating flexibility is vital to improving the cost and quality of the moldings, especially for molds with a high number of cavities or applications requiring careful control of the mold filling patterns. Second, the three-plate mold provides for the automatic separation of the feed system from the mold cavities. Automatic de-gating facilitates the operation of the molding machine with a fully automatic molding cycle to reduce molding cycle times.

There are at least three significant potential issues with three-plate molds, however. First and most significantly, the cold runner is still molded and ejected with each molding cycle. If the cold runner is large compared to the molded parts, then the molding of the cold runner may increase the material consumption and cycle time, thereby increasing the total molded part cost. Second, the three-plate mold requires additional plates and components for the formation and ejection of the cold runner, which increases the cost of the mold. Third, a large mold-opening stroke is needed to eject the cold runner. The large mold-opening height (from the top of the top clamp plate to the back of the rear clamp) may be problematic and require a molding machine with greater "daylight" between the machine's stationary and moving platens than would otherwise be required for a two-plate or hot runner mold. Given these limitations, usage of three-plate molds has declined, with an increasing usage of hot runner molds.

1.4.2 Hot Runner, Multigated, Single-Cavity Mold

Hot runner molds provide the benefits of three-plate molds without their disadvantages. The term "hot runner" denotes that the feed system is heated and so the material remains in a molten state throughout the entire molding cycle. As a result, the hot runner does not waste any material in the forming of a feed system or add any cycle time related to plasticating and cooling the material in the hot runner.

A section of a multigated single-cavity mold is provided in Figure 1.8. This mold contains a single cavity, which is designed to produce the front housing or "bezel" for a laptop or tablet computer. The hot runner system includes a hot sprue bushing, a hot manifold, and two hot runner nozzles as well as heaters, cabling, and other related components for heating. The hot runner system is carefully designed to minimize the heat transfer between the hot runner system and the surrounding mold through the use of air gaps and minimal contact area. Like the three-plate mold design, the primary and secondary runners are routed in the manifold above the mold cavities to achieve flexibility in gating locations. Since the polymer melt stays molten, hot runners can be designed to provide larger flow bores and excellent pressure transmission from the molding machine to the mold cavities. As such, a hot runner system can facilitate the molding of thinner parts with faster cycle times than either two-plate or three-plate molds, while also avoiding the scrap associated with cold runners.

During the molding process, the material injected from the machine nozzle into the hot sprue bushing pushes the existing material in the hot runner system into the mold cavity. When the mold cavities fill, the hot runner's thermal gates are designed to solidify and prevent the leakage of the hot polymer melt from inside the hot runner system to the outside of the mold when the mold is opened. The melt pressure developed inside the hot runner system at the start of the next molding cycle will cause these thermal gates to rupture and allow the flow of the polymer melt into the mold cavity.

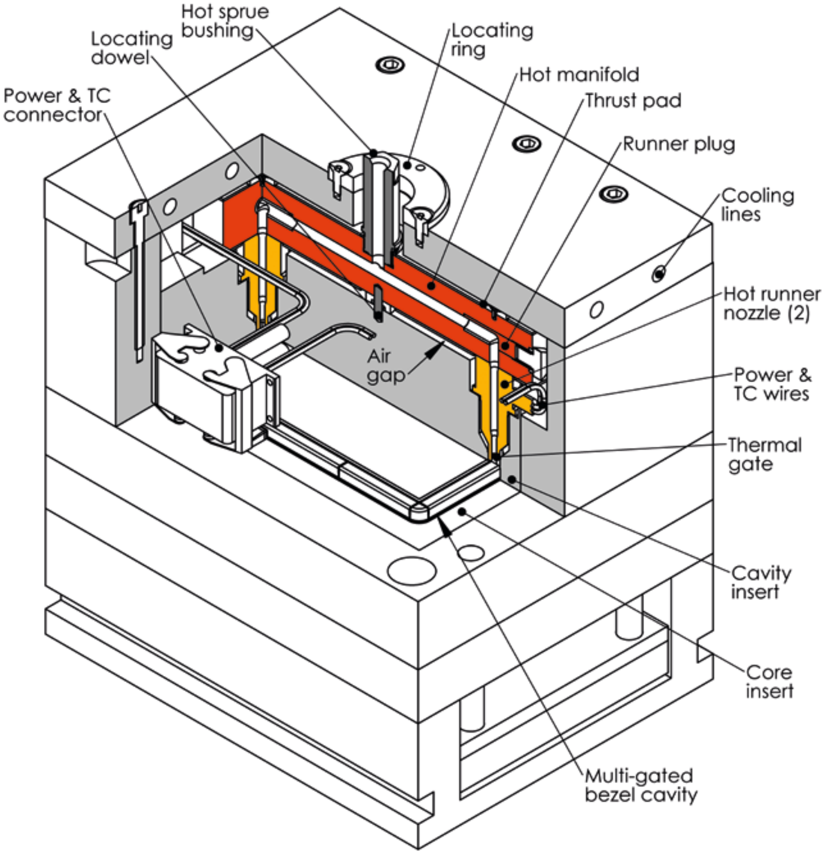


Figure 1.8 Section of hot runner mold

There are many different hot runner and gating designs that can provide advantages that include gating flexibility, improved pressure transmission, reduced material consumption, and increased molding productivity. However, there are also at least two significant disadvantages. First, hot runner systems require added investment for the provision and control of the hot runner temperature. The added investment can be a significant portion of the total mold cost, and not all molders have the auxiliary equipment or expertise to operate and maintain hot runner molds. The second disadvantage of hot runner systems is extended changeover times associated with the purging of the contained polymer melt. In short-run production applications having aesthetic requirements, the number of cycles required to start up or change resins or even color may be unacceptable. To address these issues, many molding machine suppliers offer options to integrate the hot runner controls into the machine to simplify cabling and process setup.

1.4.3 Comparison

The type of feed system is a critical decision that is made early in the development of the mold design. From a mold designer's perspective, the choice of feed system has a critical role in the design of the mold, the procurement of materials, and the mold making, assembly, and commissioning. From the molder's perspective, the choice of feed system largely determines the purchase cost, molding productivity, and operating cost of the mold.

Table 1.1 compares the different types of molds with respect to several performance measures. In general, hot runner molds are excellent with respect to molding cycle performance but poor with respect to initial investment, start-up, and ongoing maintenance. By comparison, two-plate molds have lower costs but provide limited molding cycle productivity. The evaluation of three-plate molds in Table 1.1 warrants some further discussion. Specifically, three-plate molds do not provide as high a level of molding productivity compared to hot runner molds, and at the same time have higher costs than two-plate molds. For this reason, there has been a trend away from three-plate molds with the penetration of lower-cost hot runner systems.

Table 1.1 Feed System Comparison

Performance measure	Two-plate	Three-plate	Hot runner
Gating flexibility	Poor	Excellent	Excellent
Material consumption	Good	Poor	Excellent
Cycle times	Good	Poor	Excellent
Pressure transmission	Poor to Good	Poor to Good	Excellent
Initial investment	Excellent	Good	Poor
Start-up times	Excellent	Good	Poor
Maintenance cost	Excellent	Good	Poor

1.5 The Mold Development Process

Given that there is substantial interplay between the product design, mold design, and the injection molding process, an iterative mold development process is common, such as shown in Figure 1.9. To the extent possible, the product design should follow standard design for injection molding guidelines as described in Chapter 2. To reduce the product development time, the product design and mold design are often performed concurrently. In fact, a product designer may receive a reasonable cost estimate for a preliminary part design given only the part's overall dimensions, thickness, material, and production quantity. Given this information, the mold designer develops a preliminary mold design and provides a preliminary quote as discussed in Chapter 3. This preliminary quote requires the molder and mold maker to not only develop a rough mold design but also estimate important processing variables such as the required clamp tonnage, machine hourly rate, and cycle times.

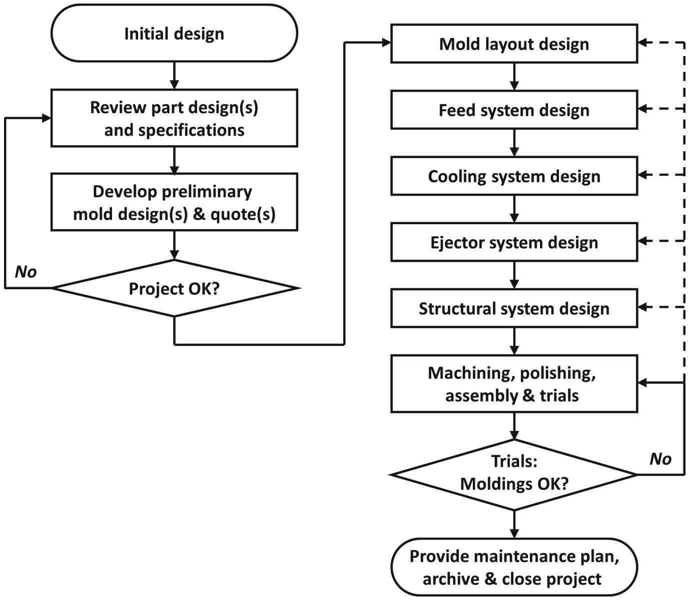


Figure 1.9 A mold development process

Once a quote is accepted, the detailed engineering design of the mold can begin in earnest as indicated by the listed steps on the right side of Figure 1.9. First, the mold designer will lay out the mold design by specifying the type of mold, the number and position of the mold cavities, and the size and thickness of the mold. Afterwards, each of the required subsystems of the mold is designed, which sometimes requires the redesign of previously designed subsystems. For example, the placement of ejector(s) may require a redesign of the cooling system while the design of the feed system may affect the layout of the cavities and other mold components. Multiple design iterations are typically conducted until a reasonable compromise is achieved between size, cost, complexity, and function.

To reduce the development time, the mold base, insert materials, hot runner system, and other components may be ordered and customized as the mold design is being fully detailed. Such concurrent engineering should not be applied to uncertain aspects of the design. However, many mold makers do order the mold base and plates upon confirmation of the order once the mold layout design is known. As a result of concurrent engineering practices, mold development times are now typically measured in weeks rather than months [4]. Customers can place a premium on quick mold delivery, and mold makers have traditionally charged more for faster service. With competition, however, customers are increasingly requiring guarantees on mold delivery and quality, with penalties applied to missed delivery times or poor quality levels.

After the mold is designed, machined, polished, and assembled, molding trials are performed to verify the basic functionality of the mold. If no significant deficiencies are present, the moldings are sampled and their quality assessed relative to specifications. Usually, the mold and molding process are sound but must be adjusted to improve the product quality and reduce the product cost. However, sometimes molds include “fatal flaws” that are not easily correctable and may necessitate the scrapping of the mold and a complete redesign. Some guidelines for mold commissioning and first article inspection are provided in Chapter 13.

1.6 Mold Standards

The designs depicted in this chapter were created from computer-aided design (CAD) files of a Milacron DME brand mold base. A “mold base” is essentially a blank mold or template design that includes all the plates, pins, bushings, and other components that may be purchased as a fully assembled system and modified for a specific molding application. Figure 1.10 depicts the prototypical mold base. This particular design [5] was made in 1944 by Ivar Quarnstrom, the founder of Detroit Mold Engineering (DME Company). It is remarkable how similar the design of Figure 1.10 is to that of Figure 1.6 and other designs commonly observed today.

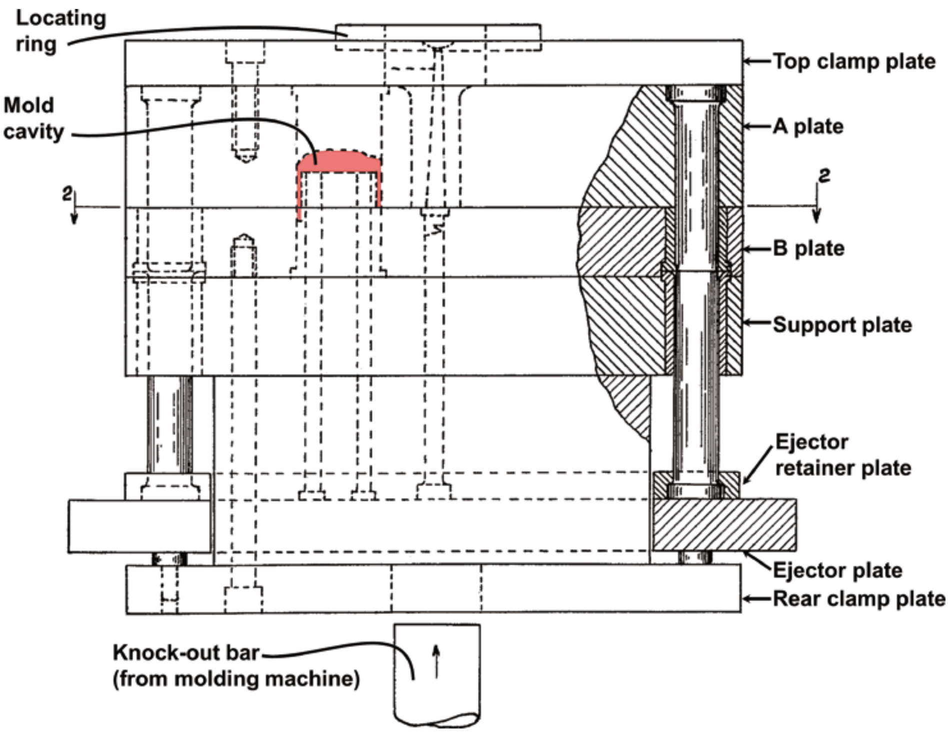


Figure 1.10 Mold base

There are many benefits for mold designs that rely on the use of the standard mold bases. These include:

- First and foremost, the design of the mold base includes many detailed fits and tolerances that would require extensive analysis and care in manufacturing. In other words, most mold designers and mold makers would have difficulty designing as good a mold base at as low a cost as a standard product that could be purchased off the shelf with minimal risk and lead time.
- Second, the use of standards provides for potential interoperability of mold bases and mold base components across molding applications as well as different molding facilities. For example, a mold designer may wish to provide six identical molds so that two copies of each mold are operable in Europe, the Americas, and Asia. The use of a mold base not only supports the mold design with respect to a CAD library, but also the provision of the replacement components using the mold base supply chain should mold components need replacement.
- Third, the use of a standard mold base provides a standard interface with typical molding machine designs. For example, consider the use of a mold base with a sprue bushing compared to a molding machine with a threaded nozzle

directly attached to a mold cavity. The use of the sprue bushing may increase the component count, but supports ready replacement and works with standard nozzle tips for a variety of molding machines. Conversely, the directly threaded nozzle eliminates the sprue altogether and so may provide better molding productivity, but requires more skill in design and operation. There is certainly the opportunity for mold designers, mold makers, and molders to outperform mold bases using custom mold designs from scratch. Such masters need significant experience and insight into their molding applications to motivate their custom designs and outperform their competitors.

For these reasons, most mold designers and mold makers in developed countries, where labor is relatively expensive compared to the mold materials and components, will typically use standard mold bases. There are many suppliers of mold bases who compete with different strategies including material technology, quality, lead time, cost, size, breadth of product line, unit systems, regional distribution, and others. Product designers, mold designers, mold makers, and molders should verify what mold base and system of suppliers are to be used in a given application.

It should be noted that the costs of fully realized molds will vary greatly, and not solely as a function of design and quality. The author has conducted research into mold quoting [6], and so is aware of instances where fully designed, machined, and finished molds have been purchased for less than the cost of just the mold base in the United States. These occurrences are often the result of inferior designs, materials, and labor practices that require extensive rework and still perform at marginally acceptable levels. With further globalization of industry, labor rates and material costs will continue to equilibrate, so product and mold designers may expect to best compete on the innovation and efficiency of their designs [7].

Adherence to standards and good engineering practices are vital to long-term competitiveness. The Society of the Plastics Industry (SPI) has provided specifications for Class 101, 102, and 103 molds intended for production of more than 1,000,000 cycles, 500,000 cycles, and 250,000 cycles, respectively. Some of the specifications are quantified. For example, Class 101 and 102 molds are required to have a Brinell Hardness Number (BHN) of 280 while Class 103 molds only require a BHN of 165. Other specifications are not quantitatively specified. For example, Class 101 molds are to have adequate channels for temperature control. Meanwhile, other specifications (like melt flow balancing and energy efficiency) are completely omitted. The engineering design and analysis methodologies presented throughout this book will assist product and mold designers to attain the best possible molds and molded products.

■ 1.7 Chapter Review

After reading this chapter, you should understand:

- the basic stages of the injection molding process,
- the primary functions of an injection mold,
- the most common types of injection molds (two-plate, three-plate, hot runner, single-cavity, multicavity, and multigated mold),
- the key components in an injection mold,
- the mold development process, and
- the motivation for standards in mold design and mold making.

In the next chapter, the typical requirements of a molded part are described along with design for injection molding guidelines. Afterwards, the mold layout design and detailed design of the various systems of a mold are presented.

References

[1] Kazmer, D.O., D. Masato, L. Piccolo, K. Puleo, J. Krantz, V. Venoor, A. Colon, et al., *Multivariate Modeling of Mechanical Properties for Hot Runner Molded Bioplastics and a Recycled Polypropylene Blend,* Sustainability (2021) 13(14): p. 8102

[2] Catanzaro, J.C. and R.M. Sparer, *Clamp force control,* U.S. Patent No. 5,149,471, Sept. 22 (1992)

[3] Poprawe, R., W. Bleck, F.T. Piller, G. Schuh, S. Barg, A. Bohl, S. Bremen, et al., *Direct, mold-less production systems,* in *Integrative Production Technology,* Springer, Cham (2017) pp. 23–111

[4] Huang, R., M.E. Riddle, D. Graziano, S. Das, S. Nimbalkar, J. Cresko, and E. Masanet, *Environmental and economic implications of distributed additive manufacturing: the case of injection mold tooling,* J. Ind. Ecol. (2017) 21(S1): pp. S130–S143

[5] Quarnstrom, I.T., *Mold Base,* U.S. Patent No. 2,419,089, filed 6/9/44 (1947)

[6] Fagade, A.A. and D.O. Kazmer, *Early cost estimation for injection molded components,* J. Injection Molding Technol. (2000) 4(3): pp. 97–106

[7] Kazmer, D.O., *Manufacturing outsourcing, onshoring, and global equilibrium,* Business Horizons (2014) 57(4): pp. 463–472

2 Plastic Part Design

2.1 The Product Development Process

Mold design is one significant activity in a much larger product development process. Since product and mold design are interdependent, it is useful for both product and mold design engineers to understand the plastic part development process and the role of mold design and mold making. A typical product development process is presented in Figure 2.1, which includes different stages for product definition, product design, business and production development, ramp-up, and launch.

How long does a product development process like that shown in Figure 2.1 take? Typically, a few months to years, depending on the complexity of the product and the number of design iterations required to develop functional components in an assembly. The most significant roadblock is the changes to the concept or layout design that impact the shape, thickness, or number of components that ripple through multiple mold design and qualification plans. To avoid such costly iterations, most product development processes share two critical attributes:

- a structured development plan [1] to coordinate concurrent design activities to ensure tracking and completeness of the design and manufacturing according to schedule and performance requirements, and
- a gated management process [2] to mitigate risk by allocating larger budgets only after significant reviews confirm expectations at project milestones.

The product development process shown in Figure 2.1 is split into multiple stages separated by approval toll-gates. An overview of each stage is next provided.

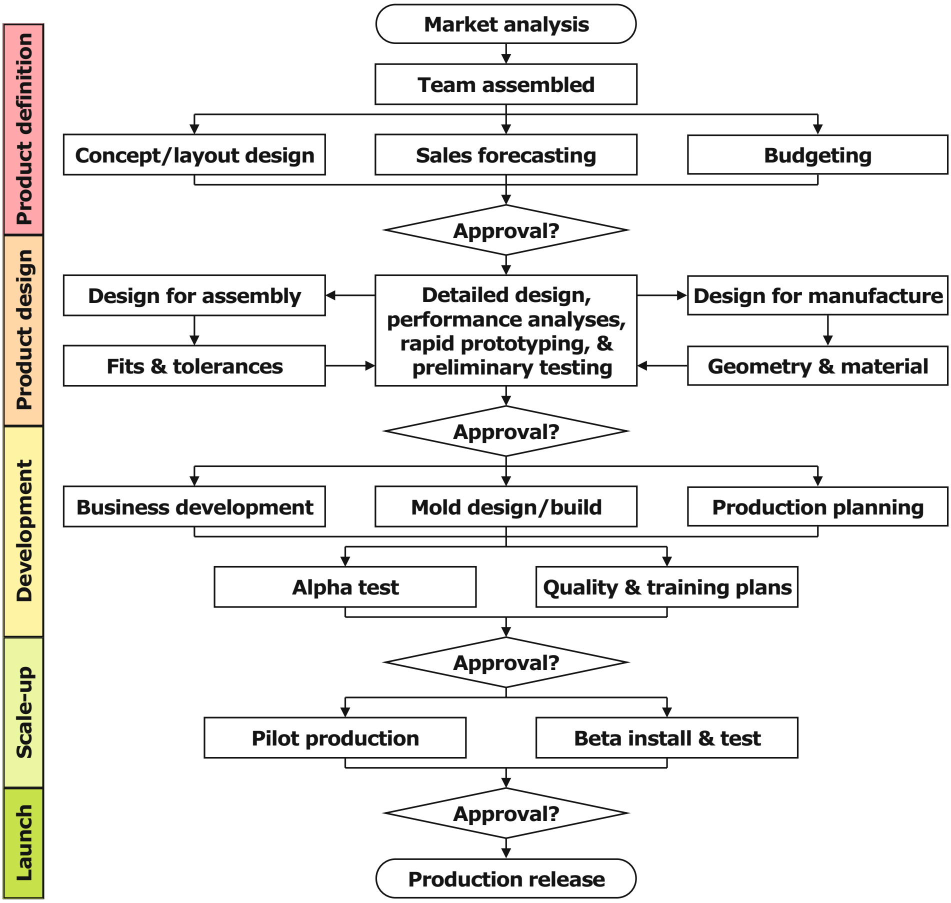


Figure 2.1 A product development process

2.1.1 Product Definition

The product development process typically begins with product definition [3, 4], which includes a formal analysis of the market, benchmarking of competitors, definition of the product specifications, and assessment of potential profitability. If management agrees that a new product is to be developed, then an appropriate team is assembled to perform the early concept design and business development. During this first stage, the approximate size, properties, and cost of the product are estimated. Concept sketches, layout designs, and prototypes are produced to assess the viability of the product concept.

With respect to profitability, market studies during the early product development stage will strive to predict the potential sales at varying price points. At the same time, labor and project cost estimates will establish the budget required to develop and bring the product to market. A management review of the concept design, sales forecast, and budget is usually performed to assess the likelihood of the commercial success of continued product development. At this time, the proposed product development project may be approved, declined, shelved, or modified accordingly.

2.1.2 Product Design

If the project is approved and a budget is allocated, then the product development process continues, usually with additional resources to perform further analysis and design. During this second stage, each component in the product is designed in detail. The design of plastic components may include the consideration of aesthetic, structural, thermal, manufacturing, and other requirements. Design for manufacturing methods [5] are used to identify issues that would inhibit the effective manufacturing of the components. Design for assembly methods [6] may be used to reduce the number of components, specify tolerances on critical dimensions, and ensure the economic assembly of the finished product.

The outcome of this product design stage (through the second management approval in Figure 2.1) is a detailed and validated product design. The term “detailed design” implies that every component is fully specified with respect to material, geometric form, surface finish, tolerances, supplier, and cost. If a custom plastic component is required, then quotes for the molded parts are often requested during this stage. These costs are presented to management along with the detailed design for approval. If the product design and costs are acceptable, then the required budget is allocated and the product development now focuses on manufacturing.

2.1.3 Development

While mold design is the focus of this book, all this content is encompassed by the single activity titled “Mold design/build” in Figure 2.1. At the same time, vital business development and production planning are being performed. Specifically, business development is required to fully define the supply chain as shown in Figure 2.2. A product manufacturer will typically work with multiple qualified molders that are typically supplied by plastic resin suppliers, mold makers, and machine suppliers. While not required, established product manufacturers often specify the suppliers to the molder to not only reduce risk but also develop strategic partnerships and potential cost or time advantages. The mold maker is a crucial supplier in this supply chain and works closely with the molder for the maintenance of the molds. However, the suppliers to the mold maker are rarely specified by the product manufacturer, though the choice of mold base supplier and mold standards are often specified in order to ensure consistent maintenance. As indicated at left in Figure 2.2, the business development must also consider the upstream supply chain including the distribution network and initial customer orders to support the product launch.

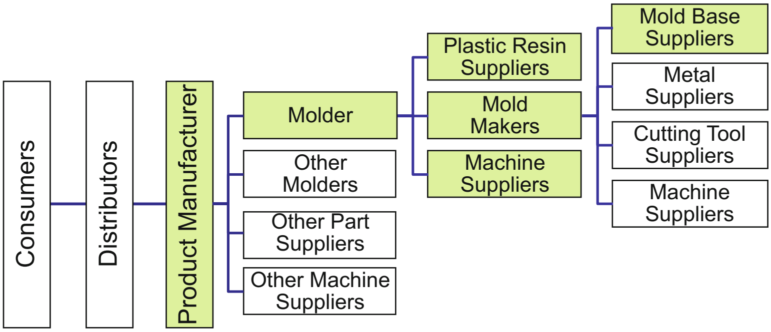


Figure 2.2 Product manufacturer supply chain

Concurrent with the mold design and making, production planning is required to lay out assembly lines, define labor requirements, and develop the manufacturing infrastructure. When the mold tooling is completed, "alpha" parts are produced, tested, and assembled. This "first article inspection" includes a battery of tests to verify performance levels, regulatory compliance, and user satisfaction. If the individual components or assembled alpha product are not satisfactory, then the manufacturing processes, associated tooling, and detailed component designs are adjusted as appropriate. Typical issues discovered at this stage include [7]:

- inappropriate performance with respect to stiffness, impact, thermal, color, assembly fits, or other attributes. These issues are often due to uncertain material properties, unidentified customer preferences that require changes to the design specification, or errors in analysis or simulation of the product performance.
- production of defective product due to mold design or tooling issues. Common examples include dimensions that are outside of specification due to shrinkage and warpage, as well as poor product aesthetics due to knit-lines, poor gating, or surface finish.
- excessive production costs related to material consumption or processing time. When quality issues are encountered, it is often possible to provide remedies through processing strategies that include increasing the temperatures, pressures, or cycle times, which then increase processing cost. Similarly, it is pos-

sible that only a fraction of the sampled products are acceptable, which results in increased material and inspection costs.

Mold designers will work with product designers and injection molders to optimize the molded product quality. Concurrently, the operations staff develops detailed plans governing quality control and worker training.

2.1.4 Scale-Up and Launch

A management review is often conducted to verify that the developed product designs and production plans are satisfactory. Prior to commercial sale, a pilot production run may be implemented at each manufacturing site to produce a moderate quantity of products to verify quality and define standard operating processes [8]. These manufactured “beta” products are frequently provided to the marketing department, sales force, and key customers to ensure product acceptability. As before, the design and manufacturing of the product may be revised to address any remaining issues. When all stakeholders (marketing, sales, manufacturing, critical suppliers, and critical customers) are satisfied, the pilot production processes are ramped up to build an initial inventory of the product (referred to as “filling the channels”), after which the product is released for sale.

2.1.5 Role of Mold Design in Manufacturing Strategy

Mold quoting, mold design, and mold making support this larger product development process. Requests for mold and/or part cost quotes are usually made towards the end of the concept design stage or near the beginning of the detailed design stage. The mold development process (first introduced in Figure 1.9) often begins with a preliminary design that is lacking in detail and would result in an unsatisfactory product if used directly. Still, the critical part design information required to begin the mold concept design includes the part size, wall thickness, and expected production quantity. Given just this information, a determination must be made as to the most appropriate manufacturing strategy as suggested in Figure 2.3 [9]. For applications requiring low production volumes, typically less than a hundred or a thousand parts, the lowest monetary and environmental costs and total production time can often be minimized through additive manufacturing [10]. Additive manufacturing is a slow and energy-efficient process compared to injection molding, however. As the production volume increases into the thousands of parts, injection molding with prototype tooling often made of CNC aluminum or other rapid prototyping materials is preferable. At large production volumes, injection molding with hardened production tooling is often most efficient.

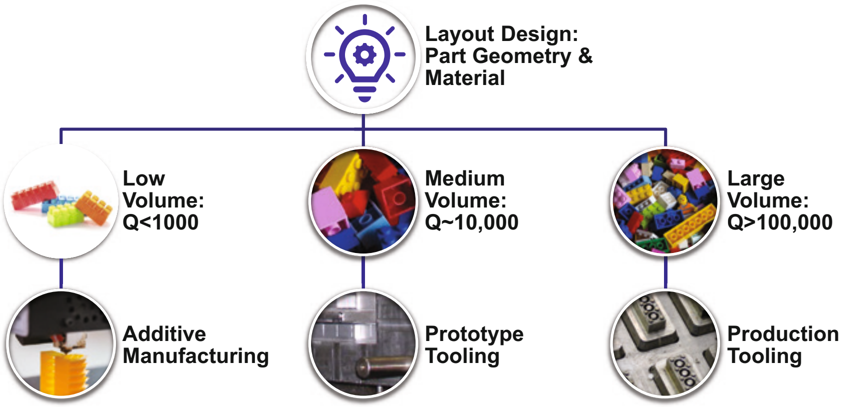


Figure 2.3 Manufacturing strategy

If prototype or production tooling are planned, the mold designer can begin to develop initial mold layouts, cost estimates, and product design improvements [11]. To accelerate the product development process, mold design can be performed concurrently with the procurement and customization of the mold components. For better or for worse, mold making and commissioning occurs near the end of the product development process. For this reason, there can be significant pressure on mold suppliers and molders to provide high-quality moldings as soon as possible. This task can be extremely challenging given potential mistakes made earlier in the product design process. As such, mold designers may be required to redesign and change portions of the mold and work closely with molders to qualify the mold for production. To minimize such issues, product prototypes can be produced to verify function as next discussed.

■ 2.2 Prototyping Strategy

Innovative products often require many iterations to design and validate. Prototypes early in the new product development effort are useful to check the feasibility of a concept with respect to size, aesthetics, stiffness, and fits in assemblies, and to validate product function. While prototype molds can be used to provide high-quality prototypes and low volume production of molded products, 3D printing by additive manufacturing has enabled functional prototypes and even low to medium volume production in many applications.

The two primary performance measures for process selection (injection molding versus additive manufacturing processes) are cost and quality. Of these two determinants, cost is easier to assess. Manufacturing services (such as Materialise, Protolabs, Shapeways, Xometry, and others) provide instant quoting for 3D printed products as well as near-instant quoting for injection molding. To investigate prototyping strategy, a bezel part design shown in Figure 2.4 is considered. The bezel is essentially a five-sided part that is roughly 240 mm long, 160 mm wide, and 11.5 mm high with a nominal wall thickness of 1.5 mm and volume of 27.5 cm^3. Internal ribs and bosses are provided for stiffening and attachment. Multiple openings are provided on different sides along with an undercutting window and transverse bosses on one of the sides.

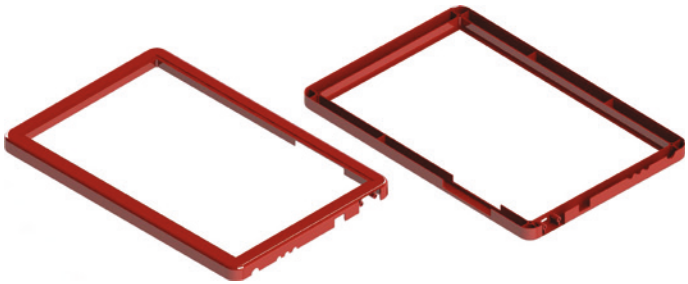


Figure 2.4 Bezel design

There are several types of 3D printing that may be used for rapid prototyping as well as cost-effective small- to medium-sized production runs. This section provides an overview of the most common processes along with cost estimates as of December, 2021. The quotes for 3D printing were provided from Xometry *(https://www.xometry.com/)* for the described bezel design that is also later used throughout the book; Xometry was selected as a representative provider of the most common processes. Other well-known services used by the author include 3DHubs, Hubs, Materialise, Protolabs, Shapeways, and others; aggregator sites such as CraftCloud also serve as a gateway to provide quotes from multiple service providers. The quote for the injection molded parts was provided by Protolabs and includes the side-action for forming the undercutting features on the side of the part. Please note that working closely with a service provider will lead to lower costs when requesting custom quotes for higher quantities of printed parts.

The quotes for producing the bezel design by the four popular 3D printing processes as well as injection molding are plotted in Figure 2.5; a semi-log scale is applied given that the production volume spans a large range. There are several important items of note. First, the cost per piece varies significantly as a function of the production volume—most spectacularly with injection molding. The reason is that there is an upfront cost for the mold, which thereafter allows for lower material and processing costs. Second, the additive manufacturing processes have quite different cost curves. Selective Laser Sintering (SLS) currently provides the lowest cost, while stereolithography (SLA) has the highest cost. The reason for these cost behaviors is related to fundamental processes and quality trade-offs as subsequently explained. Third, it is important to compare the costs with the theoretical minimum cost floor. This floor is reasonably estimated as twice the bulk material cost according to industry experience that approximates the processing, amortized tooling cost, and other costs as equal to the material cost. If the material costs \$4/kg and the part weighs 25 g, the theoretical minimum cost is around \$0.20 per part. Only injection molding can approach this cost, typically at production volumes above 100,000 units when using a well-designed mold.

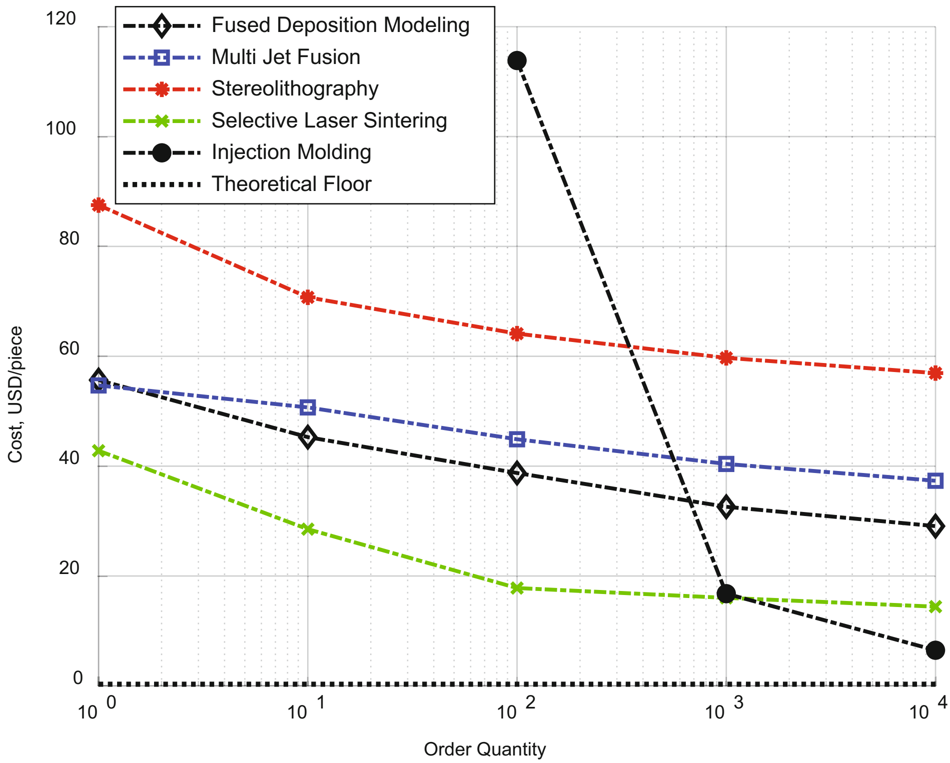


Figure 2.5 Average part production cost

The cost data of Figure 2.5 is important, but certainly production time and product quality are also significant. Clearly, 3D printing processes can provide the shortest production time for low volume production. The bezel is a medium-sized part that would take several hours to produce by 3D printing. By using many printers in parallel, it is possible to rapidly produce multiple parts. For the procured quotes, Xometry indicated 2-day delivery for up to 10 pieces and 3-day delivery for up to 100 pieces. By comparison, injection molded parts typically require 15 days given

Index

Symbols

A

E

G

H

I

T

U

V