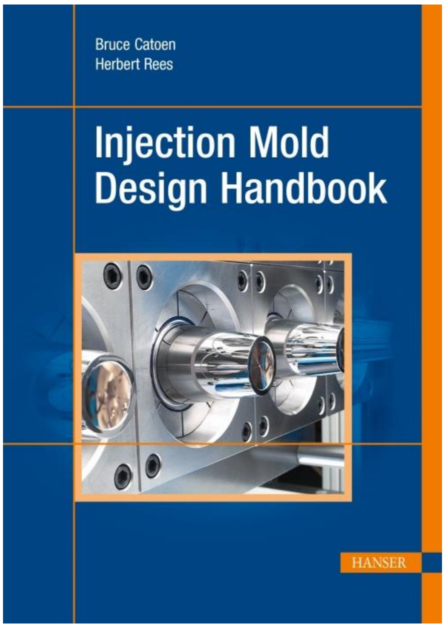
HANSER



Sample Pages

Injection Mold Design Handbook

Bruce Catoen and Herbert Rees

ISBN (Book): 978-1-56990-815-0
ISBN (E-Book): 978-1-56990-816-7

For further information and order see

www.hanserpublications.com (in the Americas)

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

Preface

From 1990 to 2005 I worked together with Herbert Rees on editing and writing books and manuals for both Husky Injection Molding Systems, where I worked in various engineering capacities, and for Hanser Publishers. Herbert was passionate about molds, design and engineering. As the VP of engineering at Husky during some of its most formative years, Herbert worked closely with Husky's founder, Robert Schad, and together they developed many machine and mold technologies.

Herbert passed away peacefully on Saturday, September 18, 2010, at the age of 95.

Over the years Herbert repeatedly told me that "An injection mold is the heart of any plastics molding work cell. Since the objective of every molder is to produce as many good parts as possible, each and every day they MUST understand the process and details of designing an injection mold". Throughout my career I have seen this proven true again and again. Understanding the principles of an injection mold design is fundamental to the success of the molded product and the molding operation.

Mold design encompasses every aspect of mechanical engineering including dynamics, statics, thermodynamics, materials, heat transfer and stress. As a result of its broad application of engineering principles it is a difficult subject to master and it results in a long learning curve for engineers. Much of the learning I received during my career was tribal knowledge based on the application of engineering principles. Gaining this knowledge took being in the right place at the right time or learning by trial and error. My hope is that this book helps short-circuit the process of learning good mold design practices.

This book is designed to be a reference handbook for the mold designer, engineer, project manager and production manager. Since designing an injection mold all starts with the plastic part, the book will first focus on key features and details of plastics and the plastic part which are necessary for good mold design. The design of the main components of an injection mold will be discussed and good design practices, rules of thumb, and key calculations will be shared. More than 600 figures, images and tables are provided in the book to illustrate how a mold should be designed. Chapter 18 contains more than 40 reference mold designs graciously provided by mold-makers around the world. These references reinforce the previous chapters and illustrate how to apply the guidelines and principles from the book into a completed mold design. Finally, the process of testing and gaining customer acceptance of the mold for production will be detailed.

By using this book as a reference guide, the reader will be able to refer to it as needed to understand:

- Critical mold design features and design practices that will ensure a successful plastic part is molded
- Detailed steps, calculations and rules of thumb for mold design
- Critical aspects of mold design such as mold layout, mold shoe design, stack contruction, cooling ejection, runner systems and materials selection
- Plastic part design requirements for a good mold design
- Processes for testing and gaining acceptance of the mold for production.

Bruce Catoen, August 2021

Acknowledgments

There are many people who helped put this book together, and I am so grateful for the time and materials that were provided to me. I would like to thank my editors and reviewers: Mark Smith from Hanser; Glenn Anderson for reviewing Chapter 5 and his contacts throughout the industry; Dr. Davide Masato, Assistant Professor at University of Massachusetts Lowell, for reviewing Chapter 11 on ejection and Chapter 12 on cooling, as well as his additional contributions on surface finishes; Fabrice Fairy for reviewing Chapter 14 on melt distribution and his boundless enthusiasm for hot runners; Vince Travaglini for reviewing Chapter 2 on part design and the tremendous help from StackTeck Systems; Vince Lomax for reviewing Chapters 9, 10, and 11, his wisdom, and the additional materials he provided on mold testing and alignment; Jordan Robertson for reviewing Chapter 5; Randy Yakimishyn for reviewing Chapters 9 and 10; Rocky Huber for reviewing Chapter 11 on ejection; Don Smith for reviewing Chapters 11 and 12 and the reference mold designs in Chapter 18; Christian Krammer for reviewing Chapter 19 and the checklists; and Dr. David Barden from Clearly Scientific Limited for his excellent edits and suggestions. Thanks to Christina Fuges for her help in connecting me with mold-makers and other industry experts to review the materials in the book.

The book would not be what it is without the generous support of the many mold-makers, hot runner suppliers, software providers, and machine manufacturers for the use of their mold designs, figures and pictures. I would like to thank: Gene Altonen CTO of IMFLUX; Paulo Silva, Managing Director of Plasdan; Mario Haidlmair, CEO of Haidlmair; Adam Chuickshank, CEO of Fourmark; Peter Smith, CEO of DME; Vince Travaglini, President of StackTeck Systems; Rui Tocha, the Director General of Centimfe; Oliver Lindenberg, VP of Global Sales Moldmasters; Andy Stirn, VP of Injection Machinery and Aftermarket NA Milacron; Mike Ellis, Global Business Manager for Hot Runners at Husky Injection Molding Systems; Miki Bogar, Senior Manager of Global Marketing at Mold-Masters; Andreas Kliber, GM of FDU Hotrunner; Stefan Von Buren, GM of MHS; Angela Vitz-Schiergens, VP Hotset; and Don Smith.

I am very grateful for all the time and effort it took to provide high-resolution drawings and figures with explanations. In particular, I want to thank: Alberto Silva at Plasdan; Fabio Och at Fourmark; Beth Thompson at DME; Jordan Robertson at StackTeck; Peter Peschl at Haidlmair; Sylvia Schmidt from Hotset; Rob Irwin at Nypro Mold; and Brenton Huxel at iMFLUX. I would also like to thank Anthony Yang and Srikar Vallury at Moldex3D for the use of figures from the *Molding Simulation* book.

Very special thanks to Jigish Shah, who provided much needed assistance with the creation of many figures for the book.

I am so grateful to Jim Sykes for his contribution of Chapter 17 on dimensioning and tolerancing.

Many thanks to Willi Miller, John DiSimone, Gord Mackay, Denis Babin, Manfred Lausenhammer, Randy Yakimishyn, and all the design and mold assembly people who took the time to teach me about mold design over the years.

Special thanks to the family of Herbert Rees for the use of materials from his books.

Finally, I want to say a very special thanks to my wife Shelley for her support, encouragement, and patience during the time it took to write this book.

The Author

Bruce Catoen, B. A. Sc.,
Mechanical Engineering,
P. Eng.

Bruce Catoen has more than 30 years of experience in the plastics industry and served as the Chief Technology Officer for Milacron and Mold Masters and as a senior executive at Husky Injection Molding Systems. Bruce is the named inventor on more than 50 patents and is author of the book *Selecting Injection Molds*. Through his consulting business, OASIC Consulting, Bruce advises senior executives on technology developments, business strategy, leadership and acquisitions. In addition to consulting, Bruce serves on two not-for-profit boards and mentors new engineers to Canada.

Contents

1 Introduction

Injection molding is a relatively new process compared to other manufacturing processes. However, in a very short period of a few decades injection molding has become one of the world's most productive and cost-effective means of producing a high-quality product.

Injection molding is a process by which plastic pellets are melted using heat and shear in an extruder and injected, at high pressure and flow, into an injection mold to form the part. While this process seems straightforward, it is full of engineering challenges and complexities.

An injection molding work cell can consist contain 4 to 16 separate elements (dryer, hopper loader, machine, hot runner, mold, robot, conveyer, etc.). Many of these elements will be standard catalog items. However, there will ALWAYS be one unique element in the work cell, and that is the mold. The mold is the **heart of the system** and all the other elements of the work cell must work together to make the unique plastic part. It is therefore critical and fundamental that the mold be designed with the utmost care and attention to detail, for if the mold does not operate as intended, then the entire work cell will operate in a subpar condition.

An old saying goes that "injection molders make money on weekends". The intended meaning is that a molder must run the first five days of the week to cover their costs, and they make profit when running throughout the weekend.

What is also written between the lines here is that injection molding is a 7-days-a-week, 24-hours-a-day business. Molders only make money when the molding work cell is producing parts. If the work cell cannot make a good part then efficiency is zero. *Molders only make money by putting good parts in the box. As a result, mold design is a critical aspect to every molded part.* Since a work cell could be in operation for many years, the mold must not only perform well on day one, but also until the last day that the production is needed.

Due to the importance of the mold design, it is critical that all levels of personnel in the molding plant and the mold-making facility understand the basics of good mold design, and the techniques used to create a mold that will allow it meet and exceed its intended purpose.

This book is therefore intended to be used, not just by the mold designer, but also by every person who comes in contact with the injection mold, so that they too can understand what makes a good mold and contribute in a meaningful way to building it.

With new, possibly difficult shapes, decisions on how to design the mold are usually left to the ingenuity of a mold designer. More frequently, *precedents* from earlier molds are used and re-applied. However, the mold designer and every person who will be involved in the molding operation must be aware of (and evaluate) new ideas, new methods, and developments, which when applied, would lead to better-quality, higher-productivity, simpler molds, and savings in the cost of the molded products.

Before proceeding with any mold design, the mold designer must understand what kind of mold should be selected. In other words, which features will be most suitable for the application to achieve the most economic *overall* manufacturing method for the product. This means not just specifying the number of cavities that will be required for the expected output, but also the *selection of mold materials and the degree of sophistication* of the mold. Any planned automation, especially in product handling *after molding*, can affect the mold layout, particularly spacing and orientation of the stacks. The mold designer must never lose sight of the ultimate goal: **to produce a part that meets or exceeds all specified requirements, at the lowest possible cost.**

The most important piece of information to know before deciding on the mold design is the quantity of parts to be molded. However, this is a piece of information, particularly with *new* products, that is often very difficult to obtain.

When looking at the overall cost of a plastic part, the per-piece cost of the mold is generally a few percent of the overall part cost (Figure 1.1 and Figure 1.2). However, the upfront cost of the mold may seem quite high. But due to the fact that it is a unique, one-off, engineered product made with very high-precision equipment to very tight tolerances by highly skilled tradespeople, the cost is realistic. On the other hand, a poorly engineered and manufactured mold is worthless, as it cannot produce a single good part.

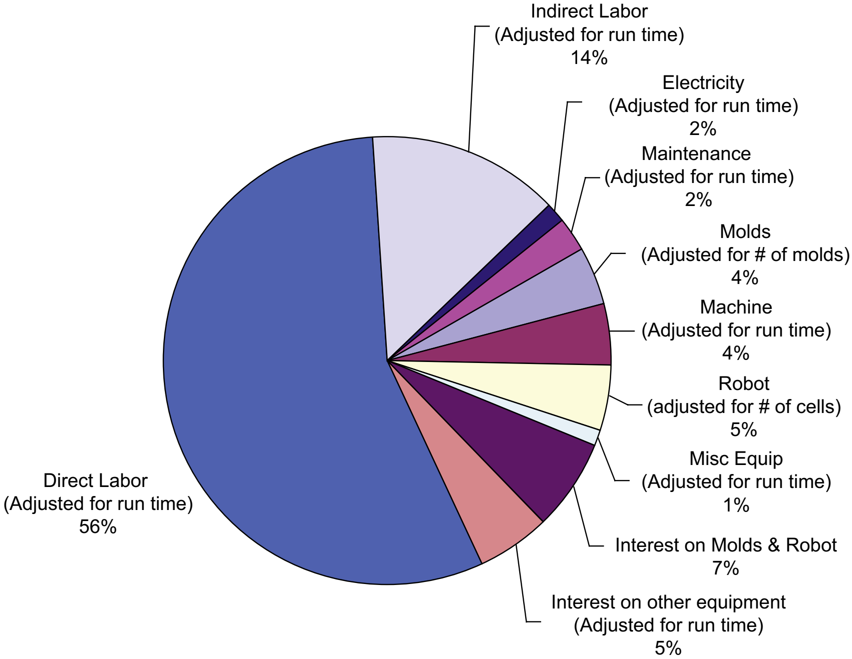


Figure 1.1 Conversion cost of an injection-molded medical part (resin excluded)

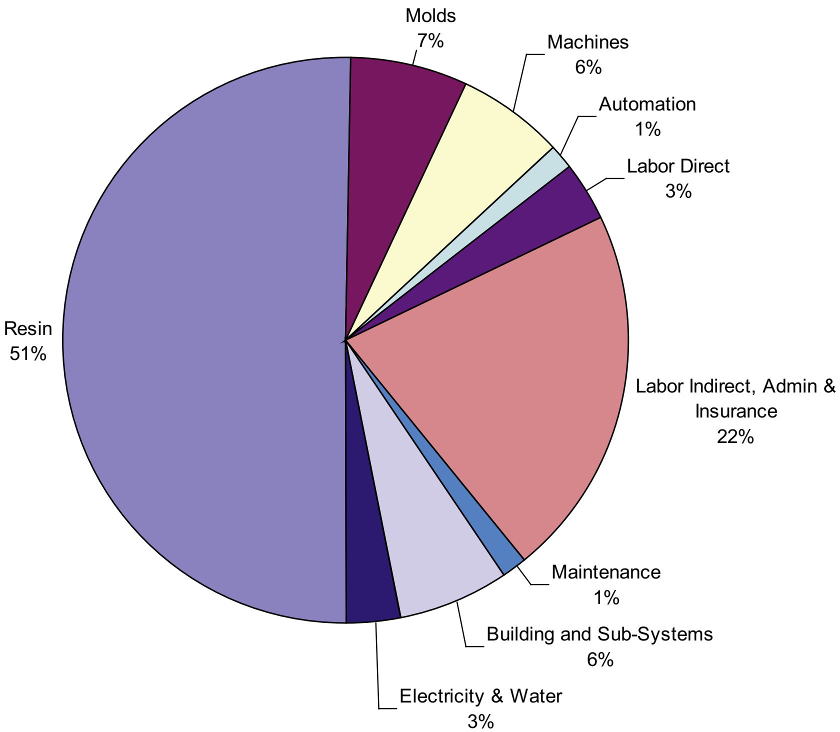


Figure 1.2 Part cost of an injection-molded pail

It should also be pointed out that of the total cost of almost all plastic products, the cost of the plastic material alone constitutes the greatest component. The most sophisticated, best-designed mold will not lower the cost of the product by as much as the reduction of just a few percent of the amount of plastic material, if it could be removed from the product without affecting its quality or serviceability. Most often, unnecessarily heavy wall thickness and ribbing affects the cost more than anything else. Chances are that the lowest weight will be achieved with the highest quality molds.

The foremost intent of this book is to present, in a logical sequence, the steps and choices available to the mold designer or decision maker when planning a mold for a new product, or when planning to increase the productivity for a product for which a mold exists. The book poses many of the questions that must be asked by anybody who needs a mold built. Any question left unanswered could significantly affect the productivity as well as the cost of a mold. For an experienced mold designer, the answers to many of these questions often come automatically, without being aware of the fact that a decision has been made. But even the most experienced mold designer can gain important information by systematically investigating all areas that can affect the design and the complexity of the mold, and checking to ensure that no obvious facts have been overlooked.

1.1 Benefits of Injection Molding

Today, injection molding is probably the most important method of processing plastics in the production of consumer and industrial goods, and is performed everywhere in the world. The benefits of using injection molding for a product or part of a product are vast and compelling. Some of the benefits are as follows:

- Low cost and high efficiency: With injection molding, a processor can produce parts in massive quantities at very low costs without high complexity or expensive skilled labor. An injection molder can set up a factory in a basic warehouse.
- Easily adapts to automated processes: Injection molding can be almost entirely automated with relative ease and low cost.
- Very high shape flexibility: Almost any shape and detail you can imagine can be injection-molded.
- Injection molding can produce parts with high tolerances and very highly detailed features or finishes.
- Excellent part properties such as light weight, high strength-to-weight ratio, excellent impact resistance, and low corrosion.

- Previously produced assemblies of multiple parts using other materials such as metal can be consolidated into one plastic part.
- Almost infinite color possibilities.
- Injection molding creates a net shape part without needing subsequent finishing.
- Plastics are easy, safe, and efficient to transport.
- Injection molding is a widely used and accepted process, so it is easy to find molders to make any part.

Before proceeding to use injection molding, the designer should always consider whether injection molding is the best solution to mold the part. Have alternative methods or product designs been considered or investigated, employing other manufacturing processes using the same or a similar materials, or using other materials which may permit a similar end product, possibly even with better quality, and/or at lower cost? A few typical examples of possible manufacturing alternatives to injection molding are:

- Thermoforming, foam molding, or blow-molding
- Coining and die-stamping (blanking)
- Machining.

The designer should also consider if other materials would be better suited to meeting the project objectives, such as:

- Paper (cardboard), wood, or cloth
- Metals (steel, aluminum, etc.)
- Glass or ceramic.

Once the decision has been made to use *injection molding* for a new product, a number of critical steps lie ahead, which will be addressed in this book:

- Plastic part design
- Factors affecting the design of an injection mold
- Mold design
- Testing and acceptance.

1.2 The Injection Mold

The heart of every injection molding work cell is the mold. It contains the form of the part that will ultimately be filled with plastic. It plays the most critical and fundamental role in the entire process – forming the part. The mold forms the desired end product. All of the other pieces of equipment in the molding work cell work to help produce a high-quality part. However, most of the other pieces of equipment in the work cell act in support of the mold.

1.2.1 The Role of the Injection Mold

Today, an injection molding work cell can contain up to 16 discrete pieces of equipment (see Figure 1.3). These devices all serve to help make a good-quality plastic part. In essence, the mold is the heart of the system, as all the other generic devices in the work cell allow the mold to make a good part. While the generic components of the work cell do not change, they must adapt every time a new mold is installed in the injection molding machine. It is therefore critical to understand the basics of an injection mold and what makes a good plastic part.

Figure 1.3 Fully automated injection molding work cell for DVD cases, containing 16 components in the work cell (Courtesy of Husky Injection Molding Systems Ltd.)

All of the supporting devices to the mold need to be correctly sized and functional in order to make the mold work well. *An injection molding work cell can be considered to be like a symphony orchestra, with the mold being the conductor. If the devices are not in tune with each other, then the whole work cell sounds like a bad high-school band, and they each contribute to making poor-quality plastic parts*. If, on the other hand, all of the supporting equipment is good working order, is correctly sized and maintained, then the resulting plastic parts will be better quality, and the work cell will run for longer without issues.

The old saying “the chain is only as strong as the weakest link” holds true in injection molding.

1.3 What Is an Injection Mold?

An injection mold is a permanent tool, i.e., a tool that, if properly designed, constructed, and maintained will have a life expectancy (useful life) *well beyond* the time where the product itself becomes obsolete. This differentiates it from a “one-time use” mold such as a sand-casting mold, as used in metal foundries. A mold can be used to make products in a virtually infinite variety of shapes, made from injectable plastics. Common to all molds is the condition that it must be possible to remove the product after molding, without the need to destroy the mold (as is the case in sand-castings).

There is an exception to this, the so-called “lost-core molding”: There are injection molds for intricate products, such as intake manifolds for internal combustion engines, previously made from cast iron, which have an outside shape that *can* be molded with conventional (permanent “open and close”) molds, but where the intricate *inside shape* is made from a molded, low-melting-point metal composite, which is inserted into the mold before injection, and then ejected together with the molded product. The metal is then removed by heat at a temperature above the melting point of the insert, but of course below the melting point of the plastic used for this product. The molded metal insert is thereby destroyed, but the metal will be reused.

A basic mold consists of two mold halves, with at least one cavity in one mold half, and a matching core in the other mold half. These two halves meet at a *parting plane* (parting line). Once the injected plastic is sufficiently cooled, the mold opens and the product can be removed by hand or be automatically ejected.

Because injection molding machines are mostly built with the injection on the stationary platen side, there is typically no built-in ejection mechanism on this side. If ejection from the injection side should be required – which is always the case in *stack molds*, and occasionally so in single-level molds – any required mechanism must be added to the mold, and occasionally to the machine; in either case, this adds complexity and increases costs. Only molds designed for using only air ejection do not require any external ejection mechanism.

Most products are removed (ejected) from the core. There are also many molds that need special provisions to allow the products to be removed from either the cavity or the core. This is the case for products with severe undercuts or recesses on the inside and/or the outside of the product, such as screw threads, holes, ribs or openings in the sides of the product, etc., or molds for insert molding.

Some of these design features of the product may require moving side cores, which are either inserts or whole sections of the cavity that move at an angle which is 90° to the "natural opening path" of the mold. Others may require special unscrewing mechanisms, either in the core or in the cavity side. The mold may require split cavities (or "splits"), i. e., the cavity consists of two or more sections, which are mechanically or hydraulically moved in and out of position, and then clamped together during injection. In some cases, the mold may require collapsible cores, or retractable inserts, which are all quite complicated (and expensive) methods.

Any of the above special features can add considerably to the mold cost when compared to a simple "up and down" mold where the products can be readily ejected with the machine ejectors during the mold opening stroke or after the mold is open, without the need for any of these complicated mold features.

Note that in this book, the term (simple) "up and down" molding is used, which comes from the earlier vertical molding machines, even though, today, most general-purpose injection molding machines are horizontal, and the mold opens and closes in a horizontal motion.

Example:

To illustrate how different mold features affect the mold cost, let a single-face mold with air ejection of the products cost X dollars. A similar mold, but with mechanical ejection, costs about $1.2X$. A similar, air-ejected two-level stack mold will be about $1.8X$. An unscrewing mold for a similar-size mold and product will cost about $2X$.

1.3.1 Elements of an Injection Mold

There are books that show designs of numerous specific molds, but it is virtually impossible to show every possible configuration that may be required. It is more important for the designer, and any person requesting a new mold, to understand that a mold consists essentially of a number of elements, from which the most appropriate for the purpose is chosen.

The reader is encouraged to read one of the following books for more a more detailed understanding of the engineering of an injection mold:

- *Gastrow Injection Molds* [1]
- *Injection Mold Design Engineering* [2].

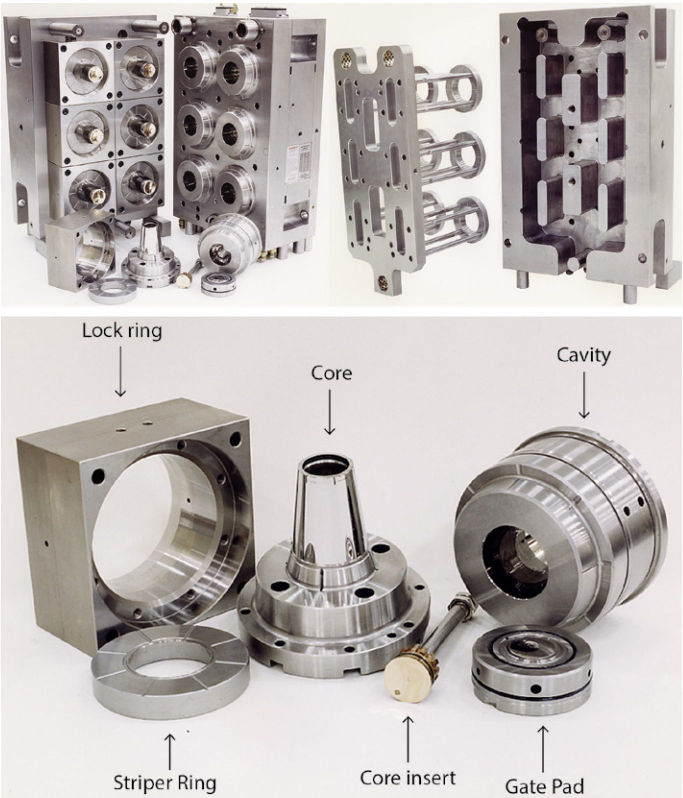


Figure 1.4 Parts of an injection mold (Courtesy of Husky Injection Molding Systems Ltd.)

[1] P. Unger (ed.), *Gastrow Injection Molds* (4th edn), Hanser Publishing, 2006.

[2] D. Kazmer, *Injection Mold Design Engineering,* (2nd edn) Hanser Publishing, 2016.

Every injection mold consists of the following basic elements (see Figure 1.4 and Figure 1.5):

1. One or more matching cavities and cores, defining the cavity space(s) (today, there are molds with anywhere between one and 256 cavities).
2. A method, or element, to duct the (hot) plastic from the machine nozzle to the cavity spaces. There is a choice between:
   - Cold runners (two-plate or three-plate systems)
   - Hot runners (various systems)
   - Insulated runners
   - Sprue gating (cold or hot).
3. Provision to evacuate air from the mold (venting). There is a choice between:
   - Natural venting
   - Vacuum venting.
4. Provision to cool the injected hot plastic sufficiently to allow ejection of the molded product.
5. Provision to eject the molded product. There is a choice between:
   - Manual product removal
   - Ejector pins and sleeves
   - Strippers (stripper rings or bars)
   - Air ejection
   - Free-drop ejection onto a conveyer
   - Various methods for in-mold product removal
   - Robotic product removal.
6. Provision to attach (interface) the mold to the molding machine. There are several methods to consider:
   - The mold is for one machine only. In this case the mold may be mounted with bolts to the platen
   - The mold is to be used on several, different machines. In this case, clamps and clamp slots on the mold may be used to bolt the mold to the platen
   - Quick mold-change methods (various designs). This could involve magnetic mounting.
7. Method of alignments of cavities and cores. There are several methods to consider:
   - No alignment feature provided in the mold. Also called flat parting line
   - Leader pins and bushings (2, 3, or 4)

- Leader pins and bushings between individual cavities and cores
- Taper fits between individual cavities and cores
- Taper fits between plates. These are also called side locks
- Any combination of the above.

8. Any number of (mold) plates to provide the necessary means for carrying and providing rigid back-up for the above elements.

In addition to the above parts, molds can have additional features, which will also be discussed in the following chapters. Each of these features can add (often considerable) costs to the mold, but in many cases they increase the productivity of the mold and reduce the cost of the product. Not all may be necessary, and each must be carefully considered when deciding on the type of mold that is most suitable (and most economical) for the job on hand.

Easy serviceability of the mold is important but often overlooked. It adds some mold cost, but saves much more in future servicing costs and downtime.

Ease of serviceability of the mold may affect the mold cost up front, but will ultimately reduce the lifecycle cost of the plastic part by reducing the need to remove the mold for service or repair. One example is the access to the hot runner for cleaning plugged gates or making minor repairs, such as changing a nozzle, a burned-out heater, or a faulty thermocouple at a hot runner drop. Building in functionality to conduct these repairs in the molding machine will cost more in the initial mold, but this will be easily recouped by reducing the downtime necessary to accomplish such repairs. By designing easy access to these components in the machine (without the need to remove the whole mold, or part of it, to the bench), such repairs can be made in less than an hour, instead of taking several hours. This work can also be done by the mold setup staff rather than getting the (expensive) mold makers involved.

Another area where valuable maintenance time can be saved is to design and provide easy access from the parting line to screws holding modular molding surface parts to their mounting plates, while the mold is in the machine. Since damage to the molding surfaces or parting lines can occur, it is advantageous to have the molding surfaces serviceable in the press. This is particularly valuable in high-cavitation molds.

Even minor changes to the part can dramatically lower or increase mold costs.

Defining what is *really* required considering the shape and complexity of the product and the required production quantities will enhance mold productivity. Alternatives and options should ALWAYS be considered and reviewed with all personnel before the mold design is finalized. Each department that interfaces with the mold will look at the design with a different set of eyes for their needs. It is of the utmost importance to include them in design reviews to ensure that the mold will have all the necessary features and functions.

Figure 1.5 shows a schematic of a basic injection mold with the key elements of the mold labelled with conventional terminology. It should be noted that the terminology used in the figure is used by the author, but there could be other names used for these components as well.

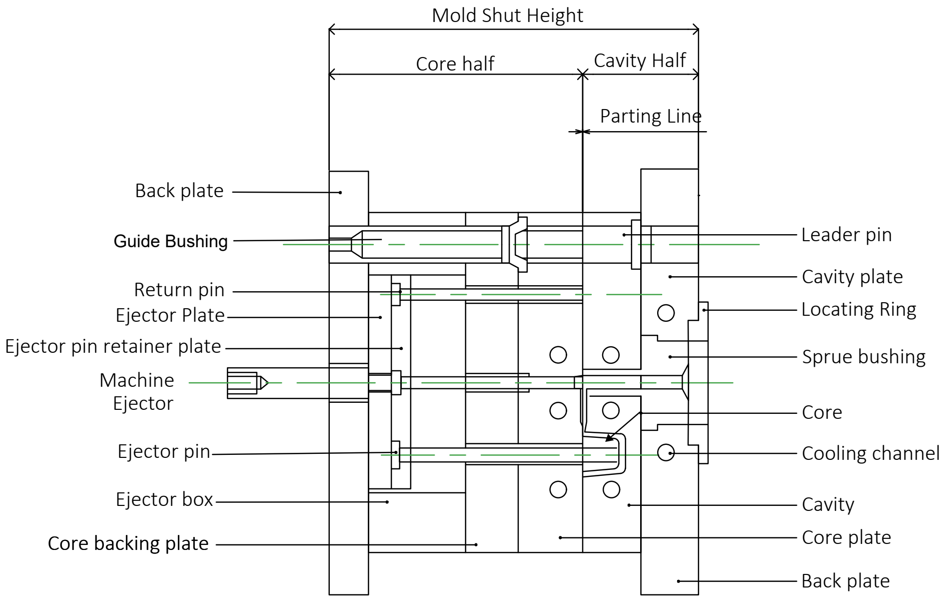


Figure 1.5 Mold terminology

1.4 Classification of Molds

SPI has developed a set of standards to classify molds by their design and intended usage. Table 1.1 explains the different standards.

Table 1.1 SPI Mold Specifications [a]

Class	Cycles	Description	Mold base	Inserts	Other
101	> 1,000,000	Built for extremely high production. This is the highest-priced mold and is made with only the highest-quality materials.	Pre-hardened 28Rc steel Stainless steel plates	All hardened > 48Rc steel Cooled inserts	Guided ejection Wear plates on slides
102	< 1,000,000	Medium to high production mold, good for abrasive materials and/or parts requiring close tolerances.	28Rc steel	Hardened steels Cooled inserts	Some guided components Some corrosion protection
103	< 500,000	Medium production mold. This is a very popular mold for low to medium production needs.	8Rc steel	> 28Rc steel	Guiding optional
104	< 100,000	Low production mold. Used only for limited production preferably with non-abrasive materials.	Mild steel or Al	Mild steel or Al	None
105	< 500	Prototype only. This mold will be constructed in the least expensive manner possible to produce a very limited quantity of prototype parts. It may be constructed from cast metal or epoxy or any other material offering sufficient strength to produce minimum prototype pieces.	Mild steel or Al	Mild steel or Al	None

[a] For more details on mold materials and the use of the Rockwell hardness scale (Rc), please refer to Section 15.2.

1.5 Continued Innovation in Molds and Hot Runners

While the use of molds dates back thousands of years, innovation continues in mold and hot runner design. There are thousands of patents on injection molding, and thousands more just on hot runners. The industry continues to innovate to provide customers with ever better ways to mold plastic products. The reader is encouraged to keep aware of the emerging trends in injection molding and to learn about how these new ideas could help to create a better injection mold. Some of the most recent trends are:

- *Conformal cooling of inserts using metal 3D printing:* Allows mold designers a additional level of design freedom in creating the cooling circuit, compared to drilling and milling multiple complex inserts and materials.
- *Direct 3D printing of plastic parts versus injection molding:* A potential threat to injection molding itself, it allows for customized creation of parts for joint replacements, running shoes, and other items that require high levels of customization.
- *Electrification of molding functions using servo motors and drives:* Functions such as rotations and stroking of pistons are now being electrified.
- *Internet connection of devices to make them "smarter",* called the industrial internet of things (IIoT or Industry 4.0).
- *Multi-material molding:* The creation of a part with multiple materials in a single process, e.g. toothbrushes, parts with integrated sealing, or parts with multiple colors.
- *Co-injection molding:* The creation of a part with multiple layers for extending shelf life, using recycled materials, and creating new aesthetics.
- *Use of gasses and liquids in the process* to core-out thick parts or to add/embed the gasses in the part for light-weighting.
- Continued development of *new resins and fillers* to create better plastics.

1.6 The Injection Molding Machine

The accuracy of molding, and especially when molding products that are difficult to produce, is very dependent on the quality of the molding machine, its mechanical rigidity, accuracy of alignment, parallelism of platens, the quality of its controls, and the state of maintenance. As mentioned previously, the equipment in a molding work cell works together in unison, and the system is only as good as its weakest link. So a high-quality mold installed in a molding machine that is poorly set up or engineered will not make a good part. The machine must be able to meet the requirements of the mold that is being installed. A good machine, poorly aligned and maintained, can destroy a new mold in a matter of months. It is imperative that the molder's machine is in good shape to ensure that the mold will perform as intended, for the lifetime intended. If this is not the case, the mold may suffer from continuous problems and issues that cannot be rectified.

There is no point in buying a premium-priced mold only to run it in an out-dated machine.

Every good injection molding machine consists of the following basic elements (see Figure 1.6):

1. A rigid base that is welded (not bolted) together using stiff box steel members.
2. A rigid clamping unit, consisting of two cast or machined platens, for the mounting of the mold halves and provisions for guiding the platens (tie bars or linear ways). The thickness of the platens is a good indicator of the rigidity and quality of the machine.
3. Provision for moving the platens, preferably fast relative to each other, for opening and closing the mold in an adjustable fashion using a fast microprocessor. Toggle-style machines tend to be the quickest machines, and fully hydraulic machines the slowest.
4. Provision for clamping, i. e., holding the mold shut against the force of the injection pressures within the mold (in some machines, provisions 3 and 4 are combined). Clamp force can be built up using hydraulic fluids or electric motors.
5. Provision for ejecting the molded product(s) from the mold. The provision can be within the mold or, more commonly, using an ejection means on the machine such as an ejector plate behind the moving platen.
6. Provision to transform the raw plastic pellets into an injectable melt. This part of the machine is called the plasticizing unit or extruder. This is almost always done using a barrel and rotating screw. The melting of the plastic is done using a combination of shear from the rotating screw and heat from the barrel heaters.
7. Provision for injecting the melt into the mold (in most machines, provisions 6 and 7 are combined in one unit). Sometimes the creation of the melt and injection of the melt are split into two separate elements. In this case the injection unit is normally called a two-stage injection unit. The injection stage is then

normally an injection or shooting pot. The injectable melt is transferred to the pot by the extruder, and a separate means then injects the melt into the mold for the shooting pot. The advantage of this more expensive approach is that the injection is more accurate and the extruder can be creating more melt while the shooting pot is injecting plastic. This can result in significantly lower cycle times.

8. Cycle controls (sequencing logic, timers, etc.) and an interface for the operator to make adjustments to the process and to operate the machine in manual or semi-automatic modes.
9. Heat controls for all heaters in machines and molds. Some machines have a limited number of heat controls, and additional controls could be required for the molds, especially with larger hot runner systems. This point must be considered when estimating the mold cost.
10. Safety gates to protect operators and bystanders from all hazards when operating the machine.
11. Mechanical safety elements to prevent closing the machine when gates are open, in case of failures of the other (electric and hydraulic) safety measures.
12. Provision for cooling water distribution to the mold.
13. Provision for compressed air, for auxiliary actions required in the mold.

There are other features available, for example, for the convenience of quick mold installation, automation, set-up and operation of the mold and machine. These features are often offered as options that can be bought with the machine or added on later.

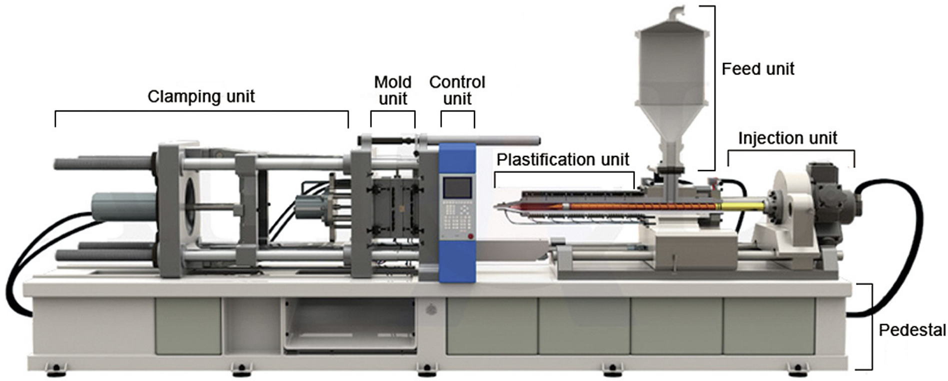


Figure 1.6 Schematic of an injection molding machine (side view) (Courtesy of Moldex3D)

Index

B

C

D

E

F

I

N

O

P

Q

R

S

T

U

V

W

Y

Z