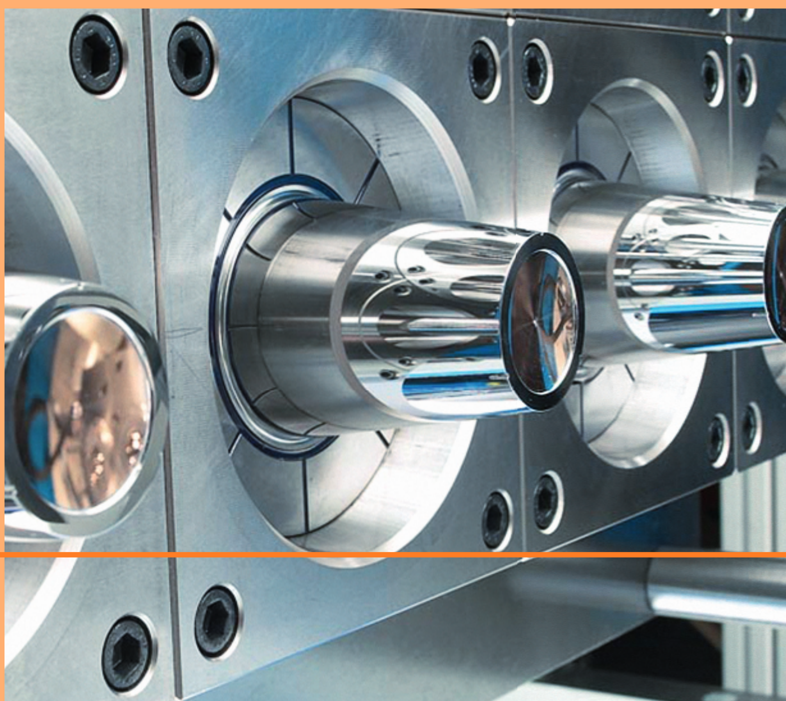
Bruce Catoen
Herbert Rees

Injection Mold Design Handbook



HANSER

Catoen / Rees
Injection Mold Design Handbook

Bruce Catoen
Herbert Rees

Injection Mold Design Handbook

HANSER
Hanser Publishers, Munich

The Authors:

Bruce Catoen, Georgetown, ON, Canada; e-mail: catb0789@gmail.com

Herbert Rees, (deceased), formerly of Mono, ON, Canada

MIX
Paper from
responsible sources
FSC® C083411

Carl Hanser Verlag
Postfach 86 04 20, 81631 Munich, Germany
Fax: +49 (89) 98 48 09
www.hanserpublications.com
www.hanser-fachbuch.de

Editor: Dr. Mark Smith
Production Management: Der Buchmacher, Arthur Lenner, Windach
Cover picture: Courtesy of Husky Injection Molding Systems Ltd
Coverconcept: Marc Müller-Bremer, www.rebranding.de, Munich
Coverdesign: Max Kostopoulos
Typesetting: Eberl & Koesel Studio, Altusried-Krugzell, Germany
Printed and bound by CPI books GmbH, Leck
Printed in Germany

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

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.

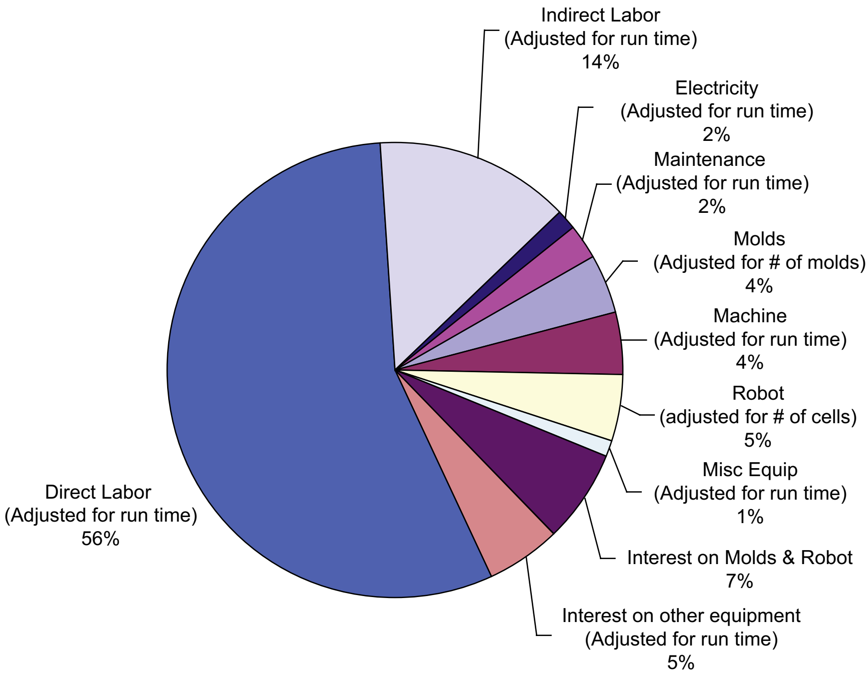


Figure 1.1 Conversion cost of an injection-molded medical part (resin excluded)

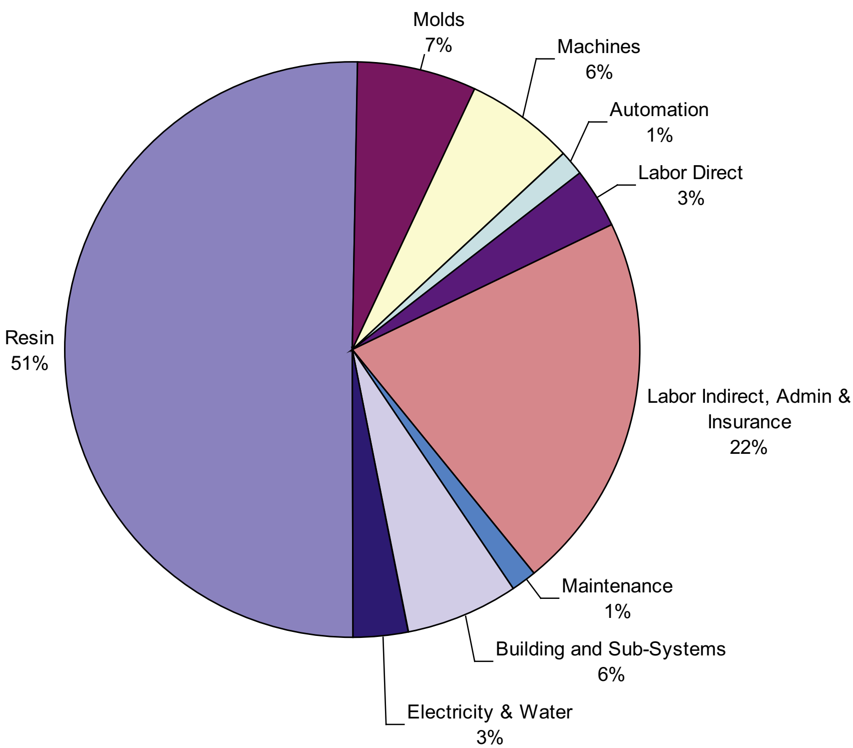


Figure 1.2 Part cost of an injection-molded pail

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].

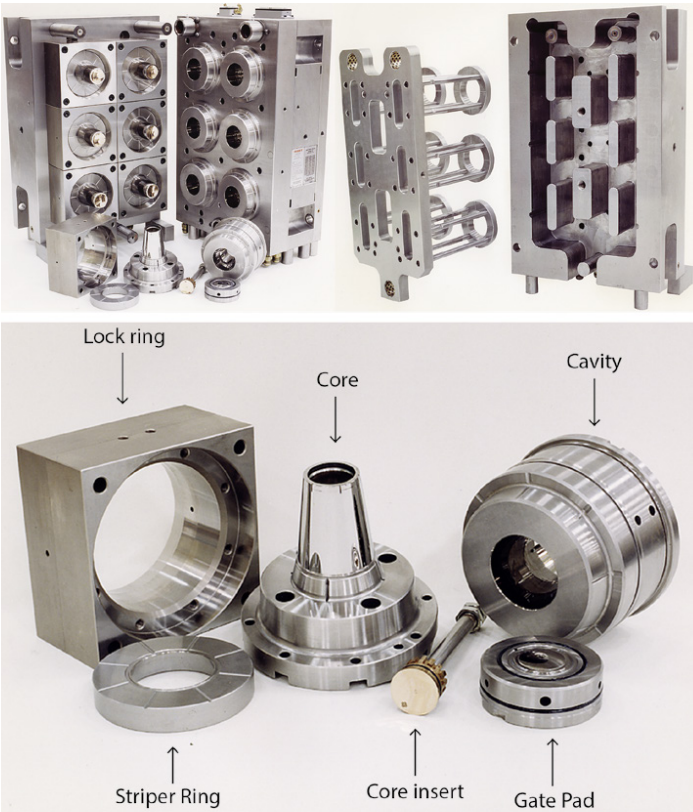


Figure 1.4 Parts of an injection mold (Courtesy of Husky Injection Molding Systems Ltd.)

[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.

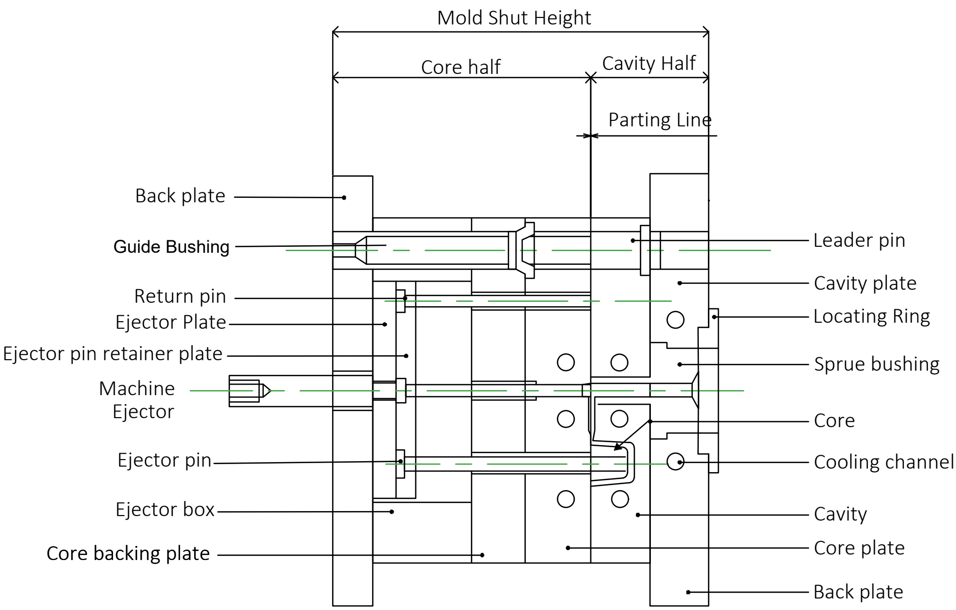


Figure 1.5 Mold terminology

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.

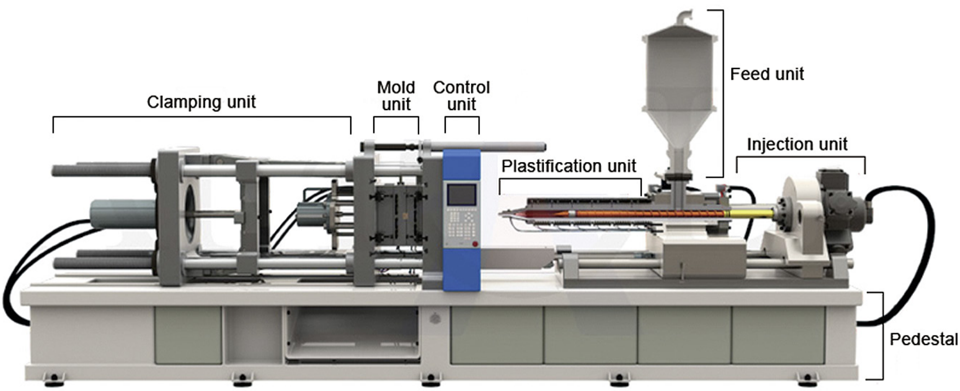


Figure 1.6 Schematic of an injection molding machine (side view) (Courtesy of Moldex3D)

2 Overview of Plastics for Mold Design

2.1 What Is Plastic?

Before considering the details of the product design, the mold designer must understand some details about the chemistry and characteristics of plastics used in injection molding.

A plastic is one of a large and varied group of materials consisting wholly or in part of carbon, hydrogen, oxygen, and other organic and inorganic elements. While solid in the finished state, at some stage in their manufacture plastics are melted (usually using heat and pressure), and thus capable of being formed into various shapes.

The process by which plastics are manufactured is called polymerization. Plastics are derived mostly from fractions of gas or petroleum recovered in the refining chemical reaction process. From these basic sources comes the monomer (a single "mer"), which is the basic hydrocarbon unit from which polymers are built. The monomer is then subjected to polymerization, by which these small molecules link together into huge molecules. Thus, the chemical reaction turns the monomer into a polymer.

A polymer may be defined as a high molecular weight distribution compound that contains comparatively simple re-occurring units. The way in which a monomer polymerizes (e.g. with chains packed close, or with chains separated by side branches) can affect its properties. Also, the size of the polymer chains and the type of molecule polymerized (e.g. substituting methyl groups for benzene rings) can affect the properties of the plastic. Polymerizing two or more different monomers together (called co-polymerization) or adding various chemicals to the basic resin can make drastic changes to the polymer's properties.

In general then, feedstocks (known as monomers) are polymerized by a chemical reaction into polymers, and the polymer is then made into useful forms for processing (e.g. granules, pellets, powder, etc.). Most plastics fall into one of two categories – thermosets or thermoplastics.

Thermoplastic resins have long molecules that are branched to one another by weak forces called van der Waals forces. Thermoplastics can be repeatedly softened and hardened by heating and cooling. An analogy would be a block of ice which can be softened, poured into a cavity of any shape, and cooled to become a solid. Examples are polypropylene, nylon, and polystyrene.

Thermoset resins consist of molecules which form cross-links with other molecules during polymerization. The chains are thus bonded together to form three-dimensional network. Once polymerized (or hardened), the material cannot be softened by heating without degrading the plastic. An analogy would be a hard-boiled egg, which has turned from a liquid to a solid during cooking, but cannot be converted back. Examples are epoxy, silicones, and phenolics. Thermoplastic resins are used almost exclusively in the injection molding industry.

There are literally thousands of thermoplastic resins commercially available today. The physical properties as well as the processing parameters used to mold these resins can vary dramatically. Appendices 8–11 list the general properties and processing conditions necessary for most of the resins used in injection molding today. These guidelines should be used for reference only: in all cases, the supplier should be contacted to obtain detailed information about a given resin.

2.2 Plastics Terminology

The following are some of the main terms used to describe the characteristics of a particular grade of injection molding resin.

Molecular Weight (Distribution)

Not all molecules are the same size. A molecular weight distribution gives the engineer a normally distributed average molecular weight. A material with a narrow molecular weight distribution (NMWD) has a molecule size which is well-controlled (i. e., a small standard deviation). On the other hand, a material with a broad molecular weight distribution (BMWD) is one which could have molecular chains of radically different sizes. Therefore, a NMWD resin will have a more consistent and usually enhanced performance compared to a BMWD resin.

Linear versus Branched Polymer

A linear polymer has no side-branching, while a branched polymer has a three-dimensional order. Note that branches of a polymer do not link with another molecule. Linking is the reaction which takes place during the molding of a thermoset resin. Linking does not occur in thermoplastic resins without degradation – a cross-linked thermoplastic resin would be basically broken down into carbon.

Homopolymer versus Copolymer

Homopolymers contain only one repeating unit, while copolymers have two or more monomer units incorporated into a single polymer. There are several types of copolymers. The second monomer and the distribution of it within the polymer can vary the material properties dramatically. ABS is a good example of a copolymer. Random copolymers show characteristics that are intermediate between the two. See Figure 2.1.

Homopolymer	–AAAAAAAAAAAAAAAAAA–
Linear copolymer	–ABABABABABABABA–
Random copolymer	–ABBAABABBABA–
Block copolymer	–AAA–BBB–AAA–BBB–
Graft copolymer	–AAAAAAAAAAAAAAAAAA– B B B

Figure 2.1 Schematic of some common types of polymers

Nucleated Polymers

Nucleating agents accelerate the formation of crystallized regions, by serving as a nucleus from which the crystallization can begin. A nucleated polymer will have many small spherulites instead of fewer larger ones. The most common nucleating agents are talc, silica, kaolin, and other polymers.

Crystalline versus Amorphous Polymers

Crystalline polymers have quite different characteristics from amorphous polymers. Table 2.1 compares the characteristics of the two materials. The materials also behave differently on heating and cooling compared to amorphous plastics.

Table 2.1 Comparison of Crystalline and Amorphous Polymers

Property	Crystalline	Amorphous
Transparent?	No	Yes
Resistance to chemicals?	Excellent	Poor
Prone to stress-crazing?	No	Yes
Propensity to shrinkage?	High	Low
Strength	High	Low
Viscosity	Low	High
Melting point	Sharply defined	Range
Volume change	Greater	Lower
Heat content	Greater	Lower

2.3 Polymer Orientation

When resin is injected into a mold, the coiled chains must untangle and orient themselves parallel to the flow axis in order to slide past each other easily. The smaller the orifice the resin has to flow through, the higher the orientation. Therefore, the gate area is usually the area with the highest orientation.

Orientation varies greatly in the cross-section of the molded part. When the material enters the cavity, the resin that hits the cold wall and freezes retains high levels of orientation. The resin at the center of the wall section can relax and shrink during the cooling stages of the cycle. Orientation increases properties in the flow direction, but decreases the properties in the transverse direction. Orientation due to differential cooling can be the major cause of plastic part warpage.

2.3.1 Shrinkage

One of the most misunderstood areas of mold design is shrinkage. Every material (metals, plastics, gases, liquids) expands as its temperature increases (heat expansion) and returns to its original volume, if cooled down to the original temperature. The problem with all plastics is the additional characteristic of compressibility. All solid materials compress under load, but most not as much as plastics. When pressure is applied to plastics (or to hydraulic oil, but not to water), plastics will compress significantly (i. e., reduce in volume) in proportion to the amount of pressure applied. This may be (within the range of molding operations) as high as 2% of the original volume.

Thus, we now have two conditions that work against each other: heat expansion and compressibility. As the plastic is injected, it is both hot and therefore expanded, but also under significant pressure, which reduces its volume. This makes it very difficult to arrive at a true shrinkage factor, because the actual change in volume depends on the type of plastic, the melt temperature, the injection pressure required to fill the cavity space, and the temperature at which it will be ejected from the mold.

Shrinkage is affected by many variables in the molding process. Some of the more important factors are:

- Type of plastic being injected
- Any additives or fillers added to the plastic material
- Heating and cooling
- Injection pressure
- Ejection temperature
- Variation of temperature across the ejected part.

For practical purposes, and for many products and molds, the shrinkage factors supplied by materials suppliers can be used. However, these figures indicate only a range within which to choose, usually between 0 and 5%. In some cases, where the volume or size of a product is important, this is not accurate enough. With crystalline plastics, such as polyethylene (PE), polypropylene (PP), and polyamide (nylon), the shrinkage factor is much higher than with amorphous plastics, such as polystyrene (PS) and polycarbonate (PC). Plastics filled with inert substances, such as glass, carbon fibers, or talcum, have a much lower shrinkage than that for the same but unfilled material. Shrinkage figures should be obtained from material suppliers, for guiding purposes.

In order to mold parts at an economically viable rate, they should be ejected as soon as possible without damaging the product. At this stage the product is still warm, and will shrink outside the mold, sometimes fast, sometimes slowly, depending on the ambient temperature and the plastic's characteristics.

There are also other considerations, such as the shape of the product. For example, if there are large differences in wall thickness or the shape is non-symmetrical, the product will cool rapidly in some areas, while remaining hot in others. This may result in warping, sink marks, or voids due to uneven shrinking once the product is ejected, because there will be still pockets of hot plastic under the cooler skin, which continue to contract, and pull on the already cooled areas. Shrinkage will also be greater in the direction of the plastic flow than across it. Due to all these factors, it is very difficult to predict shrinkages accurately in injection-molded parts. As a result, the kind of tight tolerances used in the metal industry cannot be maintained.

Mold makers generally do not guarantee the shrinkage values for the parts due to the unpredictable nature of plastic shrinkage. However, historical data can be used to guide the customer, or if necessary a prototype mold can be built. Shrinkage of height, diameter, and in the lip area may all be different. The following shrinkage sizes are commonly used but are guidelines only. The designer or project engineer should see material suppliers for more detailed and specific information for their application.

Table 2.2 General Shrinkage Guidelines (see material suppliers for detailed numbers)

Polymer	Shrinkage
PS	0.5%
PP	1.8–2.0%
PE	1.6–1.8%
LLDPE	1.5%

CAE analysis and simulation tools can be very helpful in determining where areas of high potential shrinkage may occur. The designer can then focus on this area to improve the cooling or part design to eliminate the issue. Figure 2.2 shows a simulation of a computer screen. The areas in red indicate areas of high shrinkage. The designer can then focus on these areas for improvement in the mold design before the mold is built, or they can apply a different localized shrinkage to compensate for the differences.

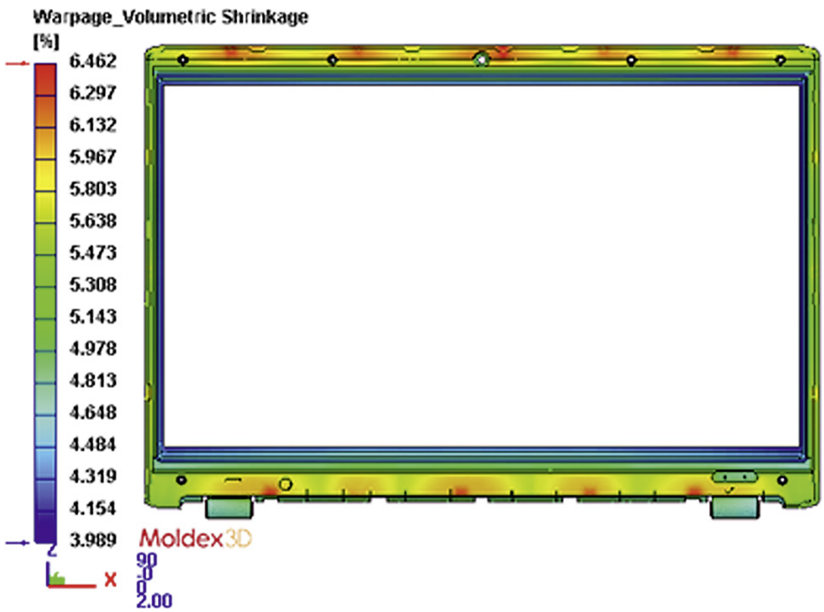


Figure 2.2 CAE prediction of shrinkage [1] (Courtesy of Moldex3D)

2.3.2 Multiple Shrinkages

The areas within the cavity spaces close to the gate see higher pressures, so the shrinkage there will be less and will require a smaller shrinkage factor. Conversely, near the end of the flow through the narrow cavity space, the pressure in the plastic is much lower than near the gate, and a higher shrinkage factor will apply. In some applications, more than two shrinkage factors may have to be selected within one cavity.

It is also important to establish at what temperature the product will be ejected. If it is ejected while still hot, it will shrink more outside of the cavity space as it cools

[1] Source: M. Wang, R. Chang, C. Hsu, *Molding Simulation: Theory and Practice,* Hanser Publishing, 2018 (Figure 3.22).

to room temperature. If ejected later, when it is cooler, it will shrink less, as measured in comparison with the steel sizes of the cavity and core. This is sometimes, but uneconomically, used to arrive at the proper size of a product such as a container or lid. If a molded product is too small because an insufficient shrinkage value was added to the product dimensions when specifying the mold steel dimensions, the proper product size can be achieved by ejecting it later, when it is cooler, but this means a loss in productivity. With high-production setups, the proper procedure is to resize the mold.

2.4 Additives

Below are some additives that are commonly added to plastics to improve some aspect of their function. As these additives can have significant effects on how the plastic behaves, it is important to know exactly what additives will be in the resin to be molded.

Plasticizers are monomeric liquids or low-melting solids of low volatility that can be used to improve the flow of the resin in the molten state. Plasticizers make the polymer more flexible, increase elongation, and lower the glass transition temperature. As a result, thinner-wall sections can be filled when these additives are used.

Fillers such as iron, copper or talc are used to increase the frictional heating during injection and improve the thermal conductivity during cooling. This results in a resin which can be molded at shorter cycles.

Lubricants are used to facilitate removal of the part from the mold, and to increase the smoothness and gloss of the molded part.

Antistats: Since plastics are excellent electrical insulators, they hold a static change very well. The surface conductivity of plastics is 1020 times greater than metals. As a result, any surface charge will dissipate slowly and attract dust and contamination. Antistats such as glycols and amines are used to combat these phenomena.

Reinforcements are used to strengthen or reinforce the properties of the virgin resin. A reinforced resin is basically a composite in which the fiber is the reinforcement and the resin is the matrix. Reinforcements used are usually very stiff and strong but brittle (e. g. glass and carbon fiber).

Colorants are used to modify the color of the molded part. Colorants are classified as either dyes or pigments. Pigments are used most often in injection molding. The color can affect the viscosity of the material, since the carrier for the pigment is usually a low-viscosity material that melts easily to allow for better dispersion of the color.

Flame retardants such as antimony trioxide are used to limit the onset of ignition and smoke generation of the resin.

Heat stabilizers are used to resist melting of the plastics. Since plastics soften and melt at a relatively low temperature they cannot generally be used in high temperature applications for extended periods.

UV stabilizers such as hydroxybenzophenes and dialkyldithiocarbonates prevent the breakdown of polymers when exposed to ultra-violet light. Ultra-violet light is absorbed by the polymer when it is exposed to sunlight. If the plastic absorbs enough energy to break the chemical bonds the resin can degrade. For outdoor applications, a UV stabilizer is required to avoid degradation.

Impact modifiers are rubbery, high-MW polymers that are added to the virgin polymer in large quantities. The result is a polymer which can resist far more shock than the virgin resin. Examples of such resins are impact-modified EVA and ABS.

Nucleating agents improve the onset of crystallinity by supplying the molten resin with a nucleus (spherulites) from which the crystal can grow.

■ 2.5 Mechanical Properties of Plastics

A detailed list of plastic properties and processing conditions can be found in Appendices 8–11.

Tensile strength (also called the ultimate strength) is the maximum load the resin can take per unit of cross-section. In tensile testing, it is the pulling stress necessary to break the specimen.

Yield strength is the lowest stress the material can handle without experiencing plastic (permanent) deformation. Plastics in general do not show a well defined yield point because of the high elongation that the polymer can withstand.

Flexural strength is the strength of the tensile bar specimen in bending. It is the maximum tensile strength the surface fibers of the resin can withstand prior to failure.

Izod impact strength is a measurement of the resistance to shock loading. It is determined by holding a notched specimen at one end, striking it, and determining the energy absorbed.

Heat deflection temperature (HDT) is the temperature at which a standard test bar of the resin deflects by a specified amount under a static load.

All of the details of these tests can be found in the ASTM standard test procedures.

NOTE: When looking at the properties of plastics, keep an open mind to accept apparent contradictions that make it difficult to pin common labels on different families of plastics. Even within various types of a single family there can be large differences. For example, low-density polyethylene (LDPE) is flexible while high-density polyethylene (HDPE) is rigid and tough (but it can also be flexible!).

■ 2.6 How Molten Plastics Behave

The study of flow and deformation of viscous fluids (called rheology) is used to determine the state of the material in the hot runner, cold runner or in the cavity. In order to understand how plastics behave when they flow (the rheology of the plastic), one must understand the principles of viscous flow relationships.

2.6.1 How Plastics Flow

Consider two parallel plates, one of which is moving with respect to the other (see Figure 2.3). This relative movement will cause a flow to be set up within the fluid. The fluid which is next to the moving plate will assume the velocity of the moving plate. The velocities of the intermediate layers will be in proportion to their respective distances from the plate at rest.

$$u = V \times (y / d) \quad \text{(see Figure 2.3)} \qquad (2.1)$$

where
V = average velocity

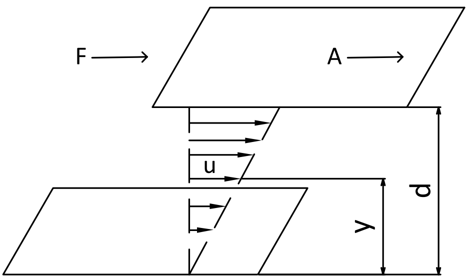


Figure 2.3
Viscous flow relationships

Isaac Newton observed that the shearing force, F, applied over the surface area A is proportional to the velocity gradient ($\mathrm{d}u/\mathrm{d}y$) within the fluid. He formulated this observation as the following equation, which shows the relationship between shear force and velocity:

$$F / A = \tau = \eta \times du/dy \tag{2.2}$$

where
η = viscosity
τ = shear stress

Viscosity is a measure of the resistance to flow. A highly viscous fluid will have a high resistance or viscosity. For example, plastics have a viscosity magnitude of 10^3, glass is 10^{20} and water is 1. Equation 2.3 shows the relationship between stress, shear and viscosity.

$$\tau = \eta \times \text{Shear rate} \tag{2.3}$$

In a round channel, *shear rate is defined as the rate of change of velocity of the moving plastic with respect to the change of radius of the round channel, and is a maximum at the wall of the runner or cavity*. The shear rate increases with the volume flow rate, but more significantly with a reduction of the melt channel diameter, as shown in Equation 2.4.

$$\text{Shear rate} = 4 \times Q / \pi r^3 \ \left(\text{in s}^{-1}\right) \tag{2.4}$$

2.6.1.1 Pseudo-Plastic Behavior

Fluids can be classified according to their flow behavior. Many common materials such as water, gas, oil, and mercury are known as Newtonian fluids (NFs). A Newtonian fluid's viscosity does not change as a function of increased shear rate. Therefore, the shear stress is proportional to the shear rate (see Figure 2.4).

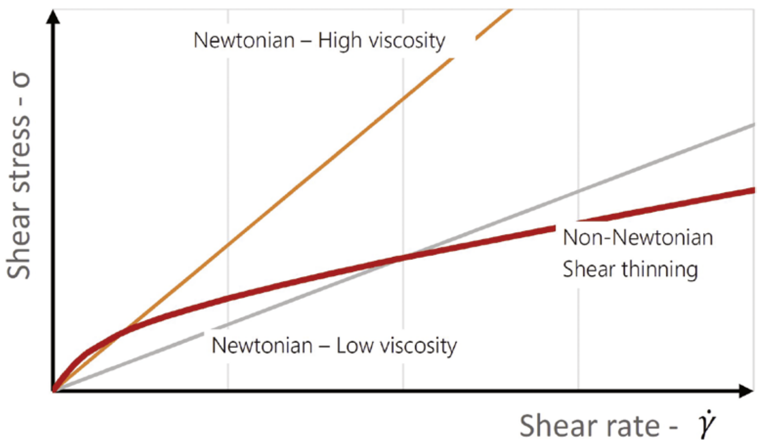


Figure 2.4 Shear stress versus shear rate [2]

[2] A. Ostergard, *Rheology Basics, www.fluidan.com/rheology-basics/*

Plastics, on the other hand, are known as non-Newtonian fluids (N-NFs). For N-NFs the viscosity changes as a function of temperature and shear rate, as shown in Equation 2.5.

$$\eta_{\text{plastics}} = f\left(\text{Temperature, Shear rate}\right) \tag{2.5}$$

In the case of plastics, as temperature and shear rate increase, viscosity decreases, but not linearly. This behavior is type of non-Newtonian fluid called a pseudo-plastic. As the shear rate increases in N-NFs, the viscosity decreases and the shear stress applied to the plastic does not increase proportional to the shear rate (see Figure 2.4). Most plastics exhibit a form of non-Newtonian behavior called pseudo-plastic behavior. For a pseudo-plastic polymer, the viscosity decreases as the shear rate increases (see Figure 2.4). This affect is called shear-thinning.

The shear–viscosity curve (Figure 2.5) is like a fingerprint of the resin because it indicates how the resin will flow at a given shear rate and a given temperature. A device called a rheometer is used to create these curves and points. However, in most cases the shear–viscosity curves are not published. Instead, an ASTM standard test (called the melt index) is conducted on the resin to determine how it flows under a given set of conditions. The melt index is an ASTM measure of the amount of plastic, in grams, that can be pushed through a defined orifice (a), at a certain temperature (b) and under a certain load (c) in 10 min. The higher the melt index the easier the material will flow. This is basically a single point on a shear–viscosity curve.

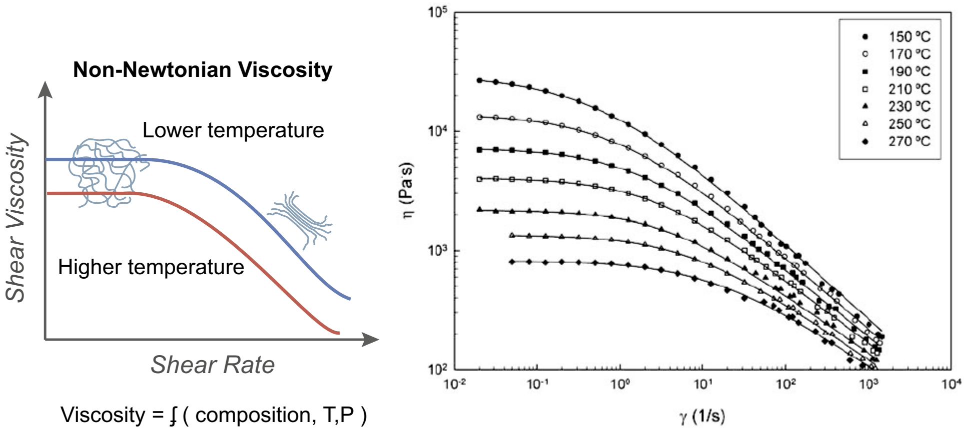


Figure 2.5 Example shear–viscosity curve for plastic

2.6.2 Plastic Flow in Runners and Cavities

Due to the behavior of plastic described in the above sections, plastic flows in a characteristic fashion called a “fountain-type flow”. Figure 2.6 and Figure 2.7 show how this looks in a cavity or runner system. Fountain flow means that the material at the center of the melt has a higher velocity than the material at the wall. This has implications for part design, mold design and melt distribution (runner) layout, as the material closest to the walls experiences higher shear rate and shear stress (thus changing its viscosity). At the same time, this material is also cooled by the core, cavity, or cold runner system. Note that the cooling effect is not present in hot runner melt channels. The impact of this type of behavior on mold design and melt flow paths will be discussed throughout the book.

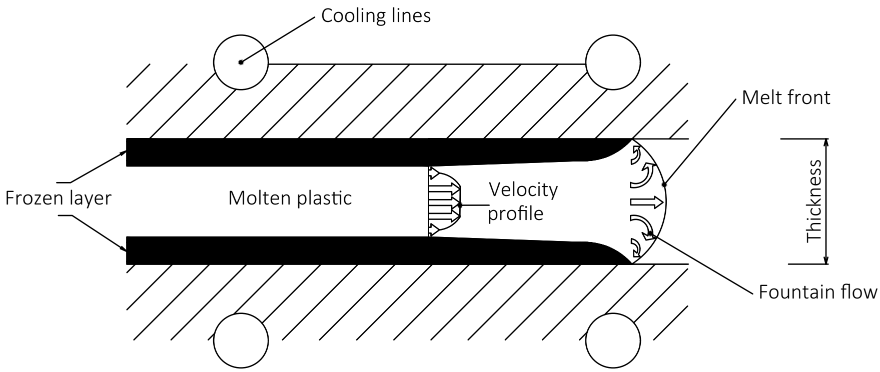


Figure 2.6 Fountain flow behavior of plastics

The impact of this flow behavior can create preferential flows in runners and in the part. In particular, when there are wall thickness changes in the part, the material will always flow along the path of least resistance. In the case of a split in the flow, like at an intersection in the runner system, the material in each sub-branch may not be identical.

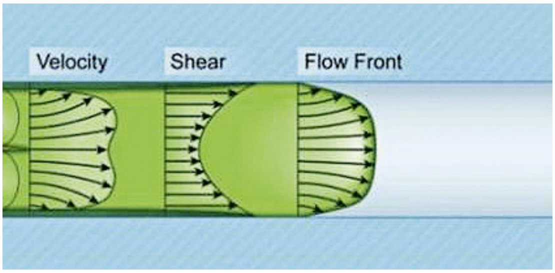


Figure 2.7 Flow in a hot runner melt channel

A good way to visualize this flow is to look at a part that has been made with two different color materials, as shown in Figure 2.8. It can be seen that the first material to enter the cavity (white) ends up on the cavity walls. The later material to enter the cavity follows the path of least resistance and ends up in the center of the part. Note that when flowing around the corner the colored material "cuts" the corner where the flow has the least resistance. This effect is also seen when looking at sections of insulated runner systems (see Figure 2.9 and also see Chapter 14 on melt distribution).

Figure 2.8 Two-color flow in co-injected plastic parts illustrates how plastic flows in the part

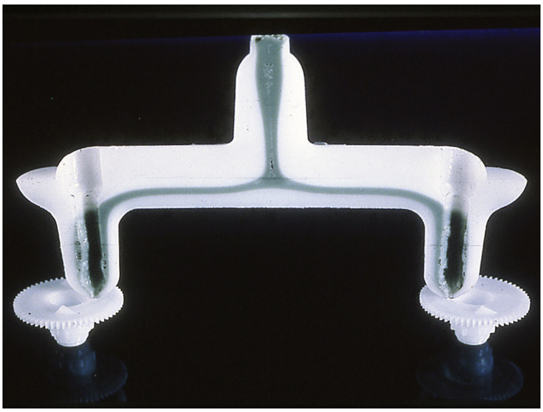


Figure 2.9 Flow in an insulated runner system showing the preferred flow path of the molten plastic

When the plastic flow splits into branches, this fountain flow behavior can cause an imbalance in filling in the runners or in the cavity itself. As the number of sub-branches or splits in the flow is increased, this imbalance in the flow is exaggerated. Eventually, if too many branches are created the flow differences may cause problems in filling all the parts equally (Figure 2.10). The impact of this behavior on the design of runner systems will be discussed further in Chapter 14.

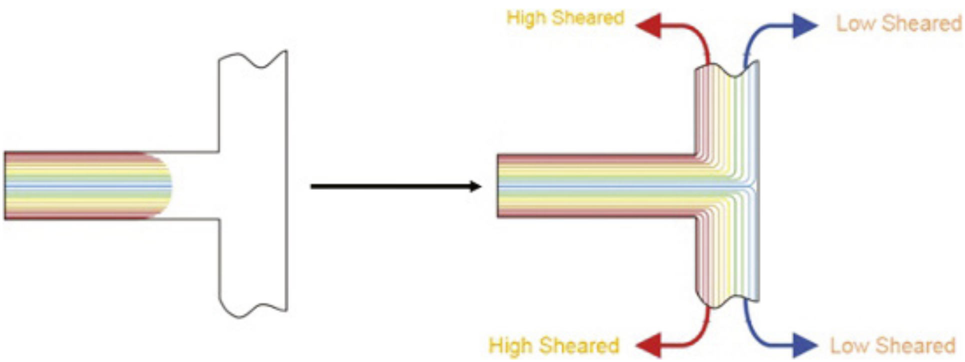


Figure 2.10 Flow of plastic at an intersection in the runner system

2.7 Degradation

Degradation is defined as any process which reduces the physical, chemical, or thermal properties of the resin. Degradation in the injection molding cycle generally occurs as the result of moisture, heat, shearing, or time. In terms of severity, heat is the most prominent, then time, shear, and moisture.

Hydrolysis or moisture degradation occurs when a hygroscopic (water-absorbing) resin comes into contact with water during the molding cycle.

Shear-sensitive materials are profoundly affected by shear. They become very easy to flow when they are sheared at a relatively low rate. This would be observed on the shear–viscosity curve (Figure 2.5) as a steeper curve. Shear-rate sensitivity can be advantageous when processing, because a faster injection rate will make the resin much less viscous and thus easier to inject.

If left above their melting point for extended periods of time, most plastics will degrade. The higher the temperature and the longer the resin is exposed to a high temperature, the more the resin will degrade or break down into its carbon base. Some materials degrade faster than others, and so must be kept in the molten state for as short a period as possible. Figure 2.11 indicates the effect of residence time and melt temperature on the time for a resin to degrade.

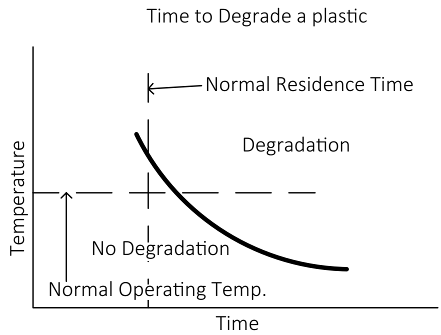


Figure 2.11 Time to degradation for a plastic

2.8 Selection and Requirements for Plastic Materials

Once injection molding has been selected as the preferred method for producing the part, the next major step in the design process will be to select the type of plastic to be used for the product. There are many criteria to consider when selecting the type of plastic to be used, and there are literally thousands of grades of plastic to choose from. The starting point for selection of the right plastic material is the requirements for the product. Table 2.3 can be used as a partial checklist for material selection. The mold designer should fully understand this before designing the mold.

Table 2.3 Checklist for Material Selection

Criteria	Range of values and/or relative importance	Options available
Color		
Weight		
Wall thickness		
Temperatures of operation		
Tensile strength required		
Compressive strength required		
Flexibility (modulus of elasticity)		
Hardness (soft or hard)		
Toughness or impact resistance		
HDT (heat deflection temperature) under load		

Table 2.3 Checklist for Material Selection *(continued)*

Criteria	Range of values and/or relative importance	Options available
Thermal conductivity		
UV resistance required		
Flame resistance		
Contacting solvents, fluids or gases		
Expected product life and reuse		
Expected material cost by weight		
Material being replaced		
Ergonomic considerations		
Surface finish or engraving?		
Product quality and appearance		
Recycling requirements		
Fillers or additives allowed		
Other		

3 Plastic Part Design for Mold Designers

Today, plastics are used in almost every area, from small bottle caps, disposable cutlery, and packages for dairy products, to large containers, such as laundry baskets and garbage pails. Plastics have transitioned from a "cheap" substitute for metal and glass to the material of choice providing almost unlimited design freedom, unique properties, and significant cost savings. Figure 3.1 and Figure 3.2 shows various durable and disposable injection-molded parts.

Figure 3.1
Molded products of various sizes (Courtesy of Husky Injection Molding Systems Ltd.)

Figure 3.2 Small and large technical (engineering) products, heavy-walled jars for cosmetics, and tubular containers with integral, hinged lids (Courtesy of Husky Injection Molding Systems Ltd.)

This chapter contains suggestions and guidelines for the *plastic product design and how it may impact the mold design and the productivity of the mold*. It is not intended to provide an extensive guideline on plastic part design, unless it materially affects how the subsequent mold may be designed. The intent is to provide suggestions on how the plastic part may be modified to ease mold design.

A new mold is usually required:

- For a new product
- If an existing product is being re-designed
- To increase the productivity and the output of the production facilities already in place. This usually provides a good opportunity to re-evaluate and improve the product, and to reduce manufacturing costs, particularly through the reduction of the plastic mass of the product. One particularly frequent productivity improvement is light-weighting of the product.

During the lifecycle of a product consideration is given to light-weighting, as it is the single biggest contributor to reducing cost. Examples of the extent of light-weighting achieved in common parts can be seen in Figure 3.3. In fact, many products are initially launched relatively heavy and then light-weighted as more is known about how the product performs.

The mass of the plastic accounts for a significant portion of the cost of every product. Reducing wall thickness and reduction of unnecessarily heavy cross-sections will not only reduce the cost of plastic material for the product, but will also result in – sometimes significantly – faster molding cycles. The result is that more of the products can be made per hour at lower cost than was possible with the preceding design. In such a case, important considerations are:

- The output of the plasticizing unit and the dry cycle of the machine manufacturing the product before the planned changes.
- If there was special handling equipment (product removal, stacking, printing, etc.) with the old mold, will it be able to handle the greater output, or will it need improvements as well?

For more in depth reading on plastic part design, the reader is encouraged to consult the following books and articles:

- *Plastic Part Design for Injection Molding: An introduction* [1]
- *Designing Plastic Parts for Assembly* [2].

[1] R. A. Malloy, *Plastic Part Design for Injection Molding: An Introduction,* Hanser Publishing, 2010.

[2] P. A. Tres, *Designing Plastic Parts for Assembly* (8th edn), Hanser Publishing, 2017.

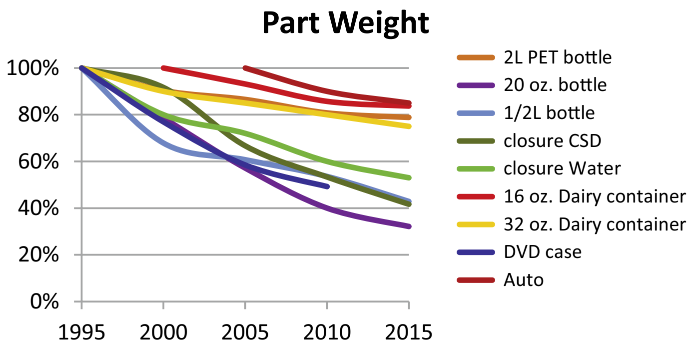


Figure 3.3 Trends of light-weighting over time in some commonly molded parts

3.1 Plastic Part Drawing

Occasionally, only samples or CAD models of a new product are available. 3D printed models are becoming more common to visualize the true part before steel is cut. This may be of some advantage to better visualize the product, but it is absolutely necessary, to minimize risk for all parties involved in the final decision, to have a complete detailed drawing of the product, showing all features, tolerances, and specifications.

It is critical that complete product drawings are available for the mold designer before any mold design (and certainly manufacturing) is started.

It is also essential that the part drawing is signed off by the end customer before the start of mold design, so that any changes in the design are noted and the corresponding change costs accounted for and billed accordingly to the customer. In today's fast-paced environment the mold designer must sometimes work in parallel with the customer by completing some of the mold design while the part is still being finalized.

An example of a complete part drawing is shown in Figure 3.4.

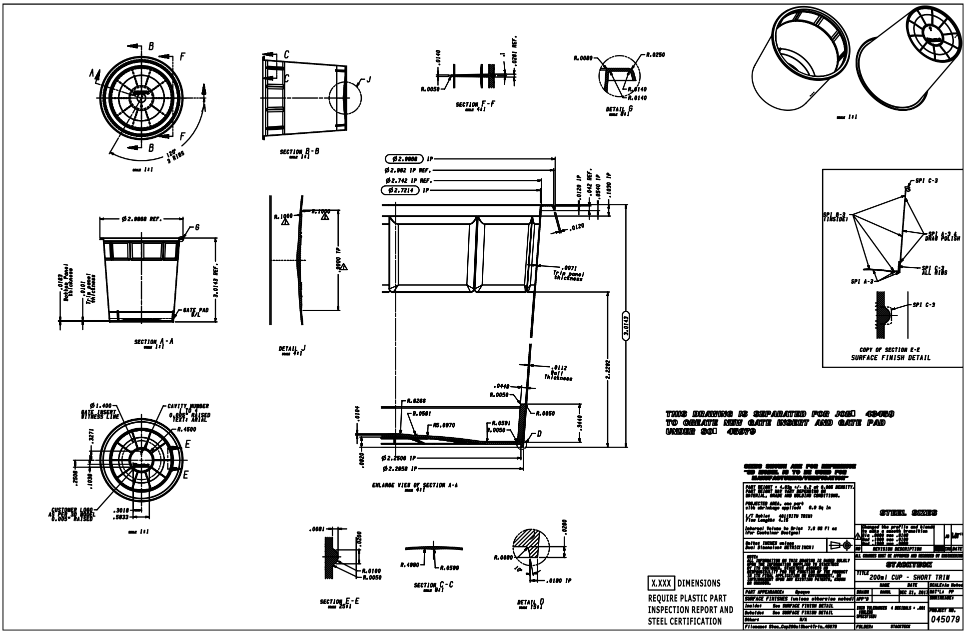


Figure 3.4 Example part drawing (Courtesy of StackTeck Systems)

A complete plastic part drawing should contain the following information at a minimum:

- A 3D solid model of the part or (at minimum) a completely dimensioned plan and section view
- Any small or intricate details blown up into additional sections or views, for example ribs or bosses
- A detail showing any fits to other components
- Engraving, artwork and cavity numbering
- Mold label information (if in-mold labeled)
- Surface finishes
- Any stacking details (if appropriate)
- Part weight (assumed plastic density)
- Part volume (if filled and weighed)
- Center of gravity (if appropriate)
- Identification of plastic or steel sizing. If plastic size dimensions are used on the drawing then shrinkage rates must be added as a note
- Identify all split lines and parting lines, including any intentional mismatches.

The checklist in Table 3.1 can be used as a part drawing critique during the part design review meeting.

Table 3.1 Review Checklist for Plastic Part Design

Checklist item	Completed (Y/N)
1. Is the drawing a plastic part drawing or "steel part drawing"? A steel part drawing is the plastic part with shrinkage dimensions applied, so that the mold designer does not need to add shrinkage. This is often used when the shrinkages are not uniform around the part.	
2. Is the shrinkage defined?	
3. Are part weight and tolerances defined?	
4. Are all drafts defined, including ribs, bosses and sidewalls?	
5. Are there no sharp corners on the drawing? Are minimum radii of at least 0.25 mm shown?	
6. Are all intentional mismatches between core and cavity shown and defined?	
7. Will the part fill? Review the L/t ratio (length of flow/thickness)	
8. Are the parting line and all split lines defined?	
9. Are locations where sinks may occur (like at the end of a rib) called out?	
10. Is the gate position defined and an acceptable gate vestige called out? Normally the acceptable vestige for a valve gate is flush with the molding surface or slightly into the molding surface to prevent interference. If the gate is a hot tip then normal acceptable vestiges are around 50–75% of the gate diameter.	
11. Is allowable warpage called out?	
12. Do the parts need to stack? If so, is the stacking height shown, and is there a diagram showing the stacking of the parts?	
13. For critical dimensions is the dimension going to be left "steel-safe" for the first run so that the sizing can be adjusted?	
14. Are all ribs shown in plan view, top view and side view?	
15. For multi-cavity molds is the cavity numbering identified?	

All sharp corners on the part drawing should be investigated and eliminated if possible. A minimum radius of 0.25 mm (0.010") should be used on plastic parts if possible. For radii bigger than 0.8 mm (0.030") the stress concentration is mostly eliminated.

While creating the plastic part drawing, the designer has the greatest opportunity to decide on the most suitable design for the mold, and/or to make suggestions on how the product design might be modified to improve the productivity, to simplify the mold design, and to reduce mold costs. This is also the time to consider any ancillary equipment required for this production. An opportunity graph (Figure 3.5) shows symbolically the value of planning a project. *Making improvements, revisions, and selections at the outset of a project offers the greatest potential to affect the final outcome of the project, including the part cost.* After concept analysis, once the elements of the project have been agreed upon and as engineering of the mold progresses, the opportunity to make conceptual changes or improvements diminishes, and any costs associated with it will increase. By the time the project reaches completion and gets into testing and production, the opportunity to make changes is low, and any costs could be very high.

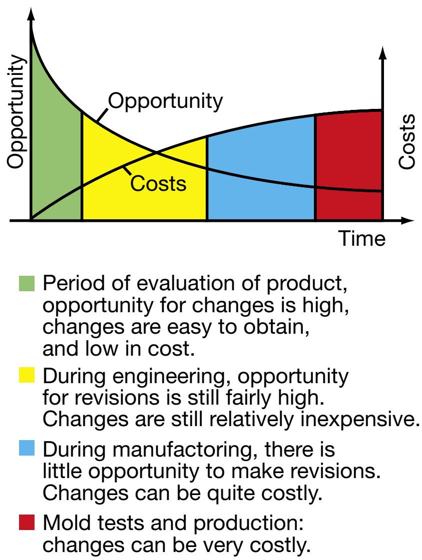


Figure 3.5 Opportunity versus cost versus time graph

■ 3.2 Product Shape: How Can the Product Best Be Molded?

A product designed for injection molding can be generally defined as flat or a box shape with some sidewalls on it. Although this is a significant oversimplification, most injection molded parts fall into one of these two categories.

The designer must consider three things right from the start (for ease of mold design):

- How will the cavity space be filled with molten plastic?
- How will the mold open?
- How will the part be ejected?

Perhaps it is the case that the answers to these three questions are obvious. If not, a designer can search the web or reference books for how a particular type of part is molded and ejected – injection molding has been around for more than 50 years, and almost every type of part has been molded. Chapter 18 of the current book provides a number of reference mold designs that the mold designer can review. A particularly good reference is the textbook *Gastrow Injection Molds* [3].

Another good way to understand how the part is made is to look at other similar parts on the market and identify the parting lines, split witness lines and ejector pin witness lines, etc. It is also advisable to consult with another knowledgeable colleague, and/or with anyone else who is familiar with the type of product for which the mold is to be built, and discuss the problems with making and of operating such a mold, to get their input regarding the proposed product design.

One of the most useful tools available for the mold designer is the use of computer-aided engineering (CAE) molding simulation software. The cost of this software is quite reasonable given the value it can provide on filling, gating, cooling, warpage, shrinkage, and more before steel is cut. The tools available from these suppliers allow the mold designer to simulate their mold concept in detail and apply molding conditions to them. They can then get a multitude of different outputs from the software that allow the mold designer to evaluate the resulting mold design, make modifications, and continue to iterate until the optimal design is found.

Figure 3.6 shows how a problem with a cellphone housing was resolved using CAE flow analysis. The existing mold was simulated on the software and compared to the actual flow-front progression of the tool. The resulting stress mark was duplicated in the software, and then modifications were made to the part until the stress concentration was removed. The tool was then modified to the recommendations made from the software, and the stress mark was eliminated.

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

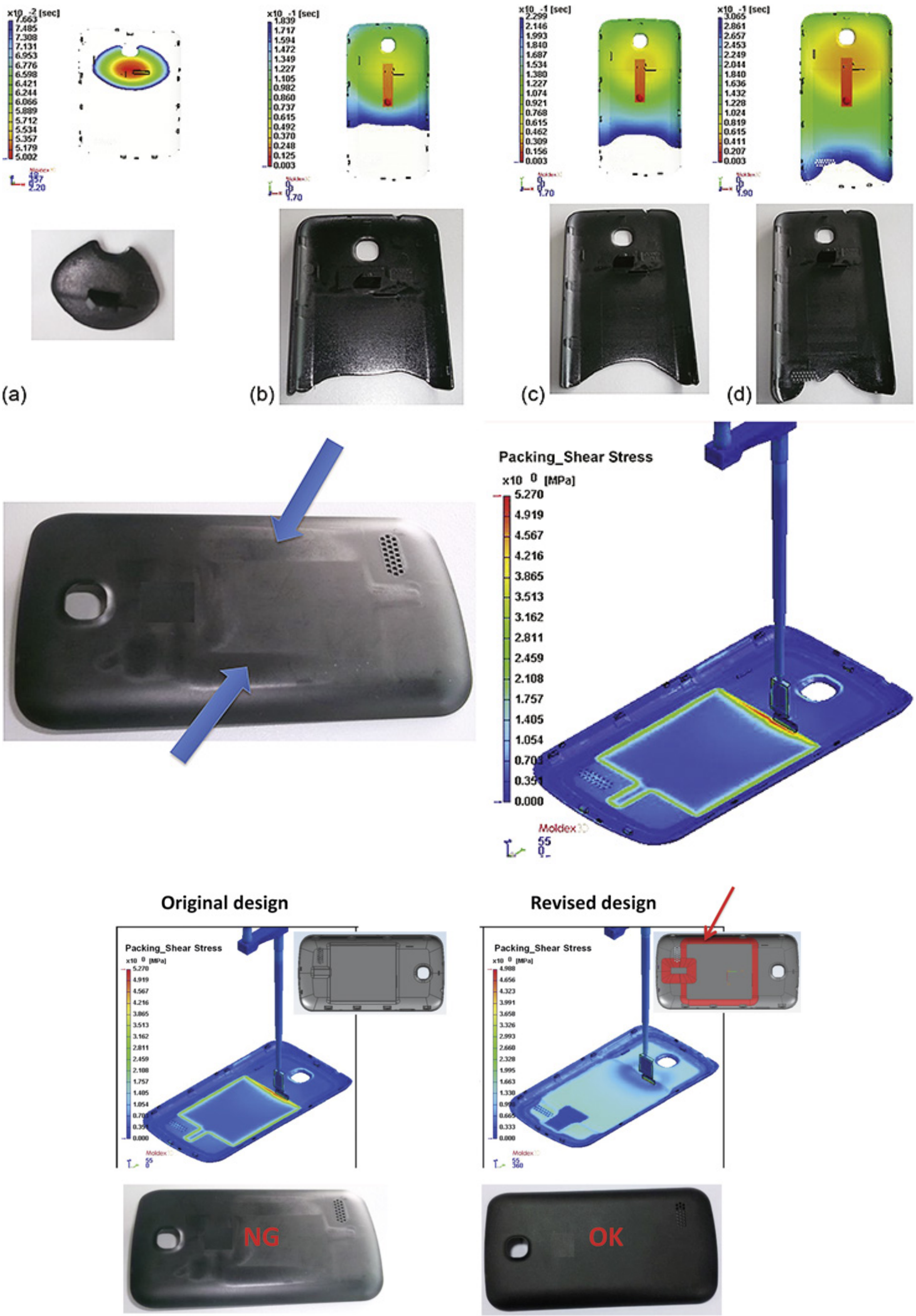


Figure 3.6 Use of computer-aided simulation tools (filling, cooling, etc.) to evaluate how to best mold a part and optimize the mold design. In this example, a stress mark on a cellphone housing was improved by comparing the progression of a mold flow front against a simulation [4]

[4] Source: M. Wang, R. Chang, C. Hsu, *Molding Simulation: Theory and Practice,* Hanser Publishing, 2018 (Figures 6.42–6.44).

3.3 Parting Line (P/L)

In many products, the location of the parting plane (usually called the parting line, P/L) is obvious. It is along the largest cross-sectional dimension of the product, at right-angles to the motion of the opening and closing of the mold, and should preferably be in *one* plane. This is the least expensive, and fortunately, the most frequent case. However, there are many cases where the P/L cannot be located there, and requires special consideration. A few examples are listed below:

- Simple parting lines (Figure 3.7).

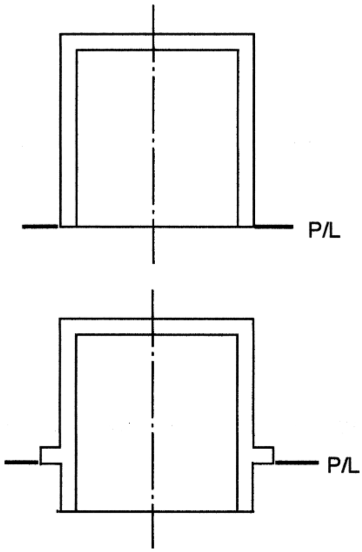


Figure 3.7 Examples of straight, simple parting lines: (Top) At the opening; (Bottom) At the largest diameter

- Sometimes, the P/L *must* be stepped or angled because of the shape of the product (Figure 3.8).

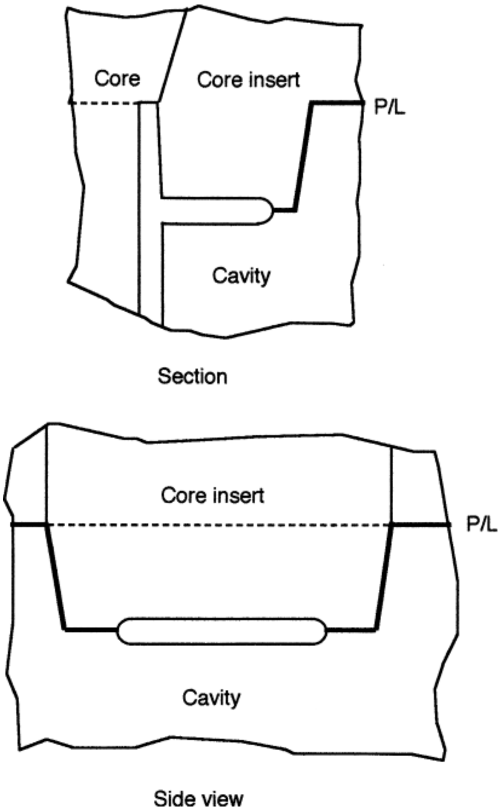


Figure 3.8 Example of simple mug handle, using an offset P/L

- It may be of advantage to place the P/L at a level that is not at the largest cross-section, to force the product to stay on the side from where it will be ejected, as can be the case with flat products. This would not affect the mold cost. However, flat products often cause trouble at ejection, because they do not always stay reliably with the side from where they are ejected. Additional mold features, such as sucker pins, or undercuts in the side of the product ("pull rings") may be required to hold the product on the ejection side, to make sure that the mold can operate automatically, without interruptions (Figure 3.9).

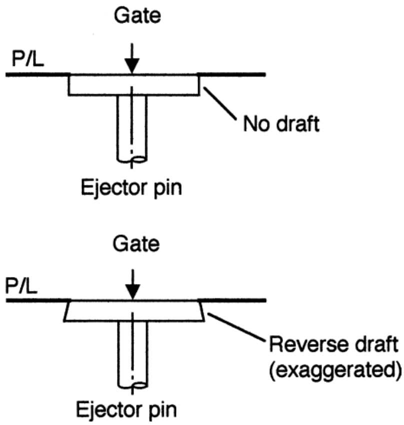


Figure 3.9 Typical flat piece with undercut below parting line to retain it to one side. Note that sometimes a change in the surface finish can cause the part to stay on the core side as well

- The P/L is curved. This is sometimes unavoidable because the product shape will not permit a straight P/L, for example in some toys, but occasionally also in technical products. A typical example is the P/L for plastic forks or spoons. In all these cases, the matching of the P/L is difficult and expensive. It may need special, costly grinding equipment or expensive fitting by hand (called "bluing") (Figure 3.10).

Figure 3.10 Typical mold profile for cutlery

3.4 Uniform Wall Thickness

An injection-molded product should, as much as possible, have a uniform wall thickness for even filling and cooling. It is better to modify the geometry of the part to eliminate thick sections such as shown in Figure 3.11A, rather than have to work out how to cool the thick sections and prevent unsightly sinks, voids and warpage. The poor design shown in Figure 3.11A will never end up looking like this in the molded part due to the changes in wall section. The better design shown in Figure 3.11B will be easier to mold and result in a part that is more consistent and has a higher-quality appearance. When considering a change from another material such as metal, wood, ceramic, or glass, the mold designer cannot just assume that this part can be replicated in plastic. The process for making the part in plastic must be considered and the part altered to take advantage of the molding process and account for its disadvantages. Decades ago, when plastics were being substituted for other materials, the biggest mistake designers made was to just try to make a duplicate of the existing non-plastic part, rather than considering how to best make the part in plastic. Once designers understood the advantages of using plastic for part consolidation, light-weighting and other features helped plastic evolve from being "cheap" to being the material of choice.

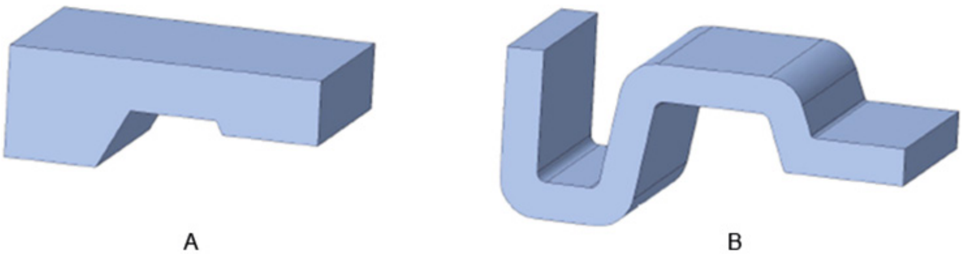


Figure 3.11 Maintain uniform wall thickness throughout the part: (A) Poor design; (B) better design

Plastic will always flow along the path of least resistance, and this must be taken into consideration when designing the part for moldability. There are a few good reasons for keeping the wall thickness as uniform as possible:

- The thicker the plastic wall section, the longer the cycle will be. The thickest portion of the molded part will control the cycle time if the part is evenly cooled in the mold.
- The thicker sections will shrink more than a thinner section, and this can cause sinks, voids and unexpected warpage in the cooled part. There are some areas where sinks are unavoidable, such as at the base of ribs and in some corners.

If differing wall sections are required, it is advisable to make the transition between thicknesses as smooth as possible and to avoid any sudden changes in wall sections, which can cause high stresses in the cooled part and very noticeable sinks or voids (see Figure 3.12 for best practice on transitions).

As a general rule, a part should be thickest near the gate and thinnest at its furthest point from the gate.

> Flowing thick to thin using the gentlest possible transitions is a general rule for plastic part design.

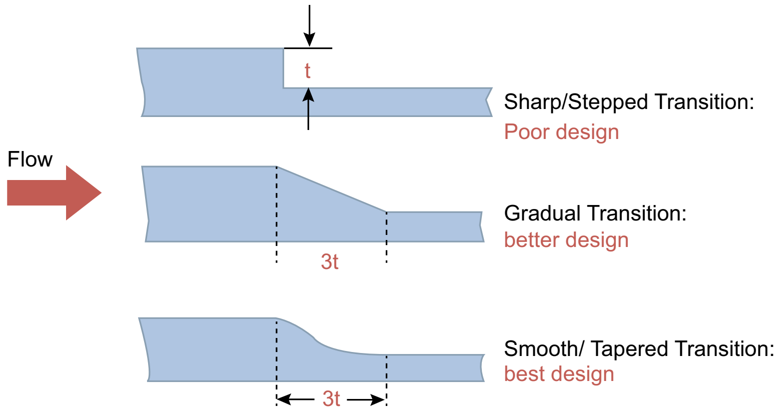


Figure 3.12 Wall thickness transitions – bad versus preferred [5]

As mentioned above, plastic will always flow along the path of least resistance, and Figure 3.13 shows how parts with uneven wall thicknesses fill. As can be seen, the plastic in the thickest areas is flowing more easily and progressing further than in the thinner areas. In some cases this can be detrimental when it causes weld lines, sinks, voids, and other molding defects. However, thicker sections can also be helpful by filling parts of the design where the plastic has further to travel.

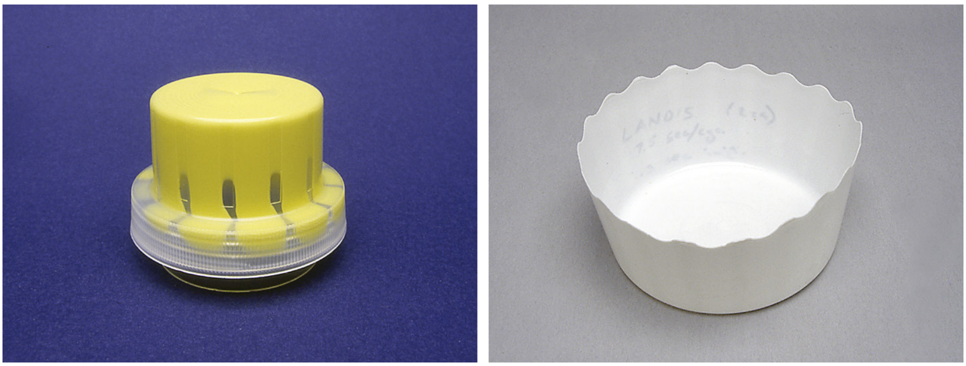


Figure 3.13 Plastic always flows along the path of least resistance (or thicker sections rather than thinner ones)

[5] Source: M. Wang, R. Chang, C. Hsu, *Molding Simulation: Theory and Practice,* Hanser Publishing, 2018 (Figure 3.3).

■ 3.5 *L/t* Ratio (Length of Flow Divided by Wall Thickness)

To keep costs down there is always the desire to make the part as thin as possible while still achieving the required part performance. In general, injection-molded parts have wall thicknesses in the range of 0.5–2 mm (0.020–0.080"). There are dramatic cases of wall thicknesses of up to 10 mm (0.4") or as low as 0.3 mm (0.010"). However, in the quest to reduce plastic consumption, the industry is pushing the boundaries of how thin a part can be molded. The most aggressive market segment in this respect is the packaging industry. Due to advances in plastic flow characteristics, as well as machinery and mold design improvements, it is now possible to mold very thin sections down to between 0.3–0.5 mm (0.010–0.020"). A good measure of how aggressively a plastic part can be thin-walled is the *L/t* ratio (length of flow from the gate to the longest end of flow, divided by the wall section. *L/t* ratios of up to 300 can be achieved in smaller parts like margarine containers using very low-viscosity plastics and up to 400 in larger parts like pails and buckets (Table 3.2).

$$L/t \text{ ratio} = \text{Flow length from gate to furthest point}/\text{Average wall thickness} \tag{3.1}$$

Table 3.2 Range of Achievable *L/t* Ratios

Part type	Resin	*L/t*
Heavy-walled parts	Low MFI [a] (PET, engineering resins)	50–100
Most parts easy to mold, high cavitation molds	Engineering resins	100–200
Higher cavitations	Commodity resins with high MFI [a]	200–300
Ultra-thin wall parts (< 0.5 mm) and very long flow-length parts with thick walls (pails)	Very high MFI resins	300+

[a] MFI is a measure of the ease of flow of a resin. It is defined as the mass of polymer, in grams, flowing in 10 min through a capillary rheometer of a specific diameter and length by a pressure applied. The higher the number the easier it will flow. ASTM 1238 and ISO 1133 standard testing methods are used to describe the required testing protocols and generate the results.

The size of the part (length of flow and wall section) and of course the viscosity of the plastic to be molded affects the achievable *L/t*. Very small parts with wall sections of less than 0.5 mm (0.020") have maximum *L/t* ratios of about 200. Much larger parts like pails, white goods, and automotive parts can have *L/t* ratios of about 400.

The number of cavities to be molded can also affect the achievable L/t ratio. Up to a 10% reduction in achievable L/t ratios can occur for large cavitations (>16).

Figure 3.14 shows sections through a molded tumbler. On the left is the original part. Note that the plastic flows from thin to thick to thin to thick, and finally to thin. This makes the part very difficult to mold, On the right is the redesigned part. Modifying the core to flow from thick to thin reduced the part weight by 20% and at the same time, decreased the cycle time by about 20%. Finally, it eliminated many molded defects caused by the thick-to-thin transitions.

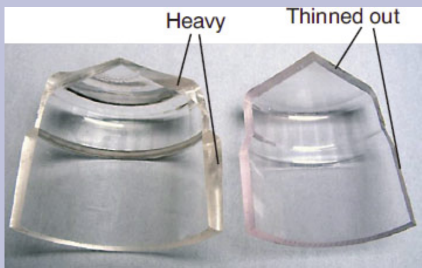


Figure 3.14 Sections through an injection-molded tumbler

3.6 Drafts

In order for a mold to open, the core must be able to open away from the cavity to eject the part. If there is no draft between the core and cavity, then the mold may "lock" during the injection process. If the design of the plastic part causes this to happen, then the opening force of the machine may be unable to separate the core from the cavity.

The following general rules for drafts should be adhered to:

- There should be drafts on all surfaces parallel to the direction of mold opening.
- Both the core and cavity should have corresponding drafts to keep wall thicknesses uniform.
- If the surface finish has significant texture, 1° of draft should be added for every 0.025 mm (0.001") of texture depth.
- Plastic part drafts must be a minimum of 0.5° for air ejection and 1° for rib drafts (0.5° per side).
- Mechanical ejection should be considered when draft angles are less than 1°.

In the event that these rules cannot be respected, then consideration must be given to more elaborate means to facilitate mold opening, such as split cavities and cores. These types of mold designs are significantly more costly and should be avoided if at all possible.

Any negative draft on the part MUST be questioned and examined. There are some cases where a negative draft might be acceptable – for instance, where a stripper ring or stripper plate will mechanically eject the part. In this case the mechanical force of ejection will cause the still-warm molded part to temporarily deform to allow the part to be ejected. Good examples of this are the undercut on a lid, stacking shoulders, and small ribs.

> Surface finishes can have a dramatic impact on whether a draft angle is sufficient for ejection. A deep etching or engraving may act as an undercut and prevent ejection without plastic part damage.

3.7 Corners, Fillets, and Chamfers

Sharp corners or fillets must be avoided in plastic part design if at all possible. Sharp corners can cause stress fractures of the part, are difficult to fill completely, increase pressures required to fill the part, and make mold manufacture more complex. As the radius or fillet gets larger, the stress concentration produced decreases rapidly. CAE analysis can help predict any stress concentrations in the proposed plastic part. These can then be addressed before the mold is designed.

The corner radii should match to produce an even wall section throughout the corner, as shown in Figure 3.15. If the outside radius is too large, then the filling can be restricted and the part weakened. If the outside radius is too small, then the part becomes too thick in this region, causing localized racetrack filling of the part and wasted plastic in the part.

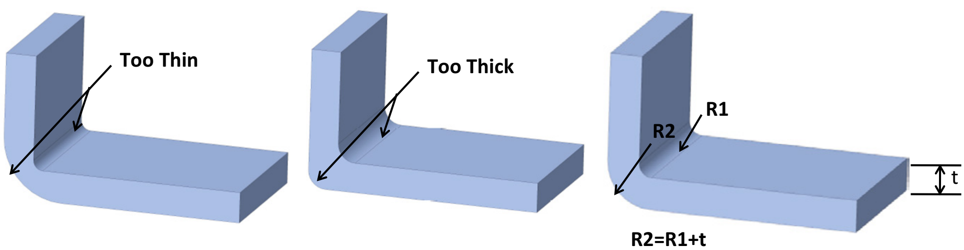


Figure 3.15 Corner design

As a rule of thumb, any external corner or radius should be a minimum of 150% of the wall thickness, and the corresponding internal radius should be at least 50% of the wall thickness.

3.8 Ribs and Bosses

Ribs and bosses are commonly used in plastic parts for reinforcement and for assembly. While needed for the physical strength of the molded part, they can cause molding defects (such as sinks voids, shorts, and trapped air) if incorrectly specified. To prevent defects in the molded part, the following guidelines should be followed for ribs and bosses:

- Rib height should be less than 3 × the part wall thickness at the rib.
- The draft on all ribs and bosses should be 0.5° or more per side.
- The span between two ribs should be > 2 × the wall thickness at the rib.

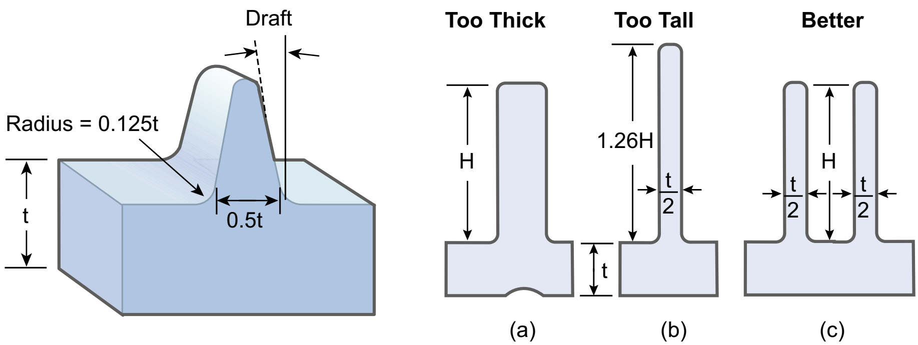


Figure 3.16 Recommended dimensions for a rib [6]

[6] Source: M. Wang, R. Chang, C. Hsu, *Molding Simulation: Theory and Practice,* Hanser Publishing, 2018 (Figures 3.11 and 3.14).

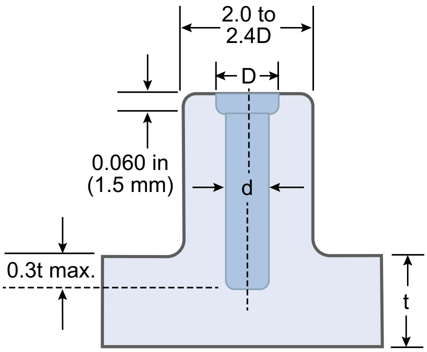


Figure 3.17
Recommended dimensions for a boss [7]

A good way to hide the location of a rib or a boss is to put a transition in this area so that the sink is not noticeable (Figure 3.18).

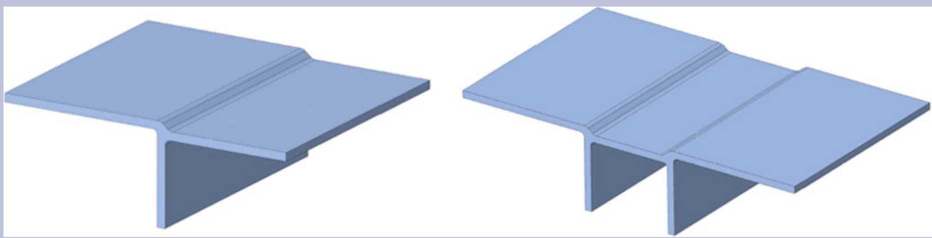


Figure 3.18 Hiding the location of a rib or boss

A rib should not end in a sharp corner, as this makes the rib difficult to fill. Instead a radius or chamfer should be added to the end of the rib to make filling easier (see Figure 3.19). The exception to this guideline is when the rib is being used as a feature, and the top of the rib surface area is needed for something like stacking or alignment with another part.

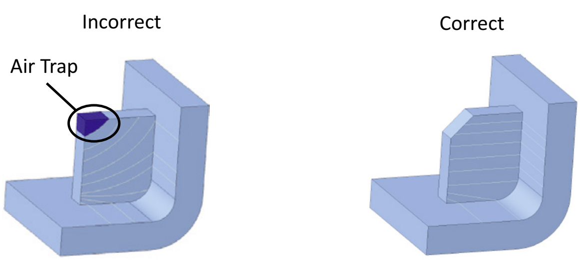


Figure 3.19 Avoid sharp corners at the end of ribs

[7] Source: M. Wang, R. Chang, C. Hsu, *Molding Simulation: Theory and Practice,* Hanser Publishing, 2018 (Figure 3.16).

3.9 Rim Designs

Figure 3.20, designs A–C, shows round rim designs frequently specified on plastic parts. The large bead rim shape (Figure 3.20A) is not recommended. As the rim is at the end of the flow path, the injection pressure is greatly reduced, and such a rim will be difficult to fill and pack. The result is either a void, usually with amorphous plastics such as PS which freeze rapidly on the surface (Figure 3.20A1), or a sink, usually with crystalline plastics which freeze slower and pull the still-soft plastic skin toward the center, as shown, exaggerated, in Figure 3.20A2 and shown in Figure 3.21. The bead rim dimension will be difficult to control dimensionally and will cause sealing issues or fit issues with the mating component. A better plastic part design would avoid such large cross-sections at the end of any flow path and the mold maker should explain it to the product designer, and request a change as suggested in Figure 3.20B or C which will cycle much faster and not have the risk of sinks or voids. An additional benefit of designs B and C is that these options use less plastic and permit faster cycles, resulting in lower product costs.

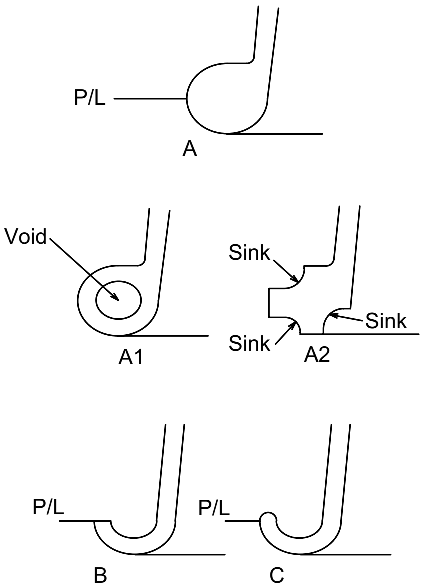


Figure 3.20
Beaded rim, with void and sinks

Figure 3.21
Cut section of the rim of a pail with a large void

■ 3.10 Stripped Undercuts

To strip an undercut, the lead angle (see Figure 3.22) needs to be between 30° and 45° in order that the plastic can travel up the incline and out of the undercut. The surface finish of the part in this area needs to be smooth and highly polished to avoid any hang-up points. The size of the undercut to be stripped depends on the flexibility of the resin and the total surface area of the undercut. The undercut itself can usually be in the range 0.35–0.75 mm in depth per side (that is $D - d = 0.7$–1.5 mm). More detail on stripper ejection of undercuts can be found in Chapter 11.

Certain resins that are stiff, such as PS, PC, nylons and other reinforced resins should not be stripped. Instead these parts should have split cores or cavities or other motions that free the undercut before ejecting the part.

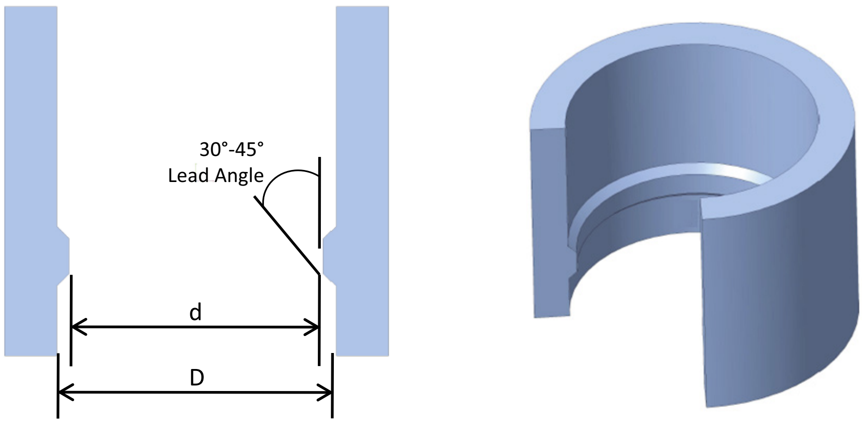


Figure 3.22 Stripped undercut should have a generous lead-in, smooth radii and polished molding surfaces

■ 3.11 Sidewall Windows in the Part

Sometimes the plastic part requires openings on the sidewall of the part. This presents a challenge in mold design, as these openings need to be filled with steel during injection, and then the steel needs to retract from the opening during mold opening and ejection. One way to create these holes is to use a side motion in the mold that occurs during opening and ejection (see Chapter 11 on ejection). However, if the plastic part is designed with this in mind, the expensive side motion may be avoided. Figure 3.23 shows an example of where the shape of the window