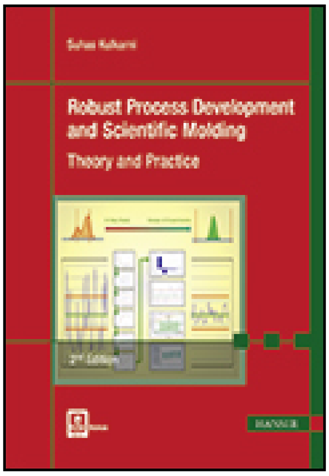
HANSER



Sample Pages

Suhas Kulkarni

Robust Process Development and Scientific Molding

Theory and Practice

Book ISBN: 978-1-56990-586-9

eBook ISBN: 978-1-56990-587-6

For further information and order see

www.hanserpublications.com (in the Americas)

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

Contents

Preface to the Second Edition

As the saying goes "the only thing that is constant is change." It has been six years since the first edition of this book was published, and it has been very well received. Thank you to all its readers. Since then, I have continued my research to further understand the process of injection molding with the final goal of robust process development. As I kept publishing and teaching this new material, it became time to revise the book.

This second edition has new material in almost all the chapters. Some concepts, which were explained in the first edition, have been expanded upon and rewritten for better understanding. Several figures have been added to complement the explanations. Some of the chapters and text have been split up and rearranged to have a better flow of understanding. A complete chapter on "Basic Quality Concepts" has also been added.

The topic of process development is a complex one, but once the concepts are understood, implementation is easy. The key is to understand the basics first. Over the years, in my consulting business, I often get called on by companies to 'fix' their processes. I always go back to the basics and ask them several simple questions about their molds, machines, and processes to which they sometimes have no answer, or when they do answer my questions, they figure out the solution to the problem on their own. Their process development was probably done by throwing darts on a dartboard and hence the issues. This book is attempting to change that. By using the techniques described in this book, one can establish what I call *cruise control processes*: set the process, start molding, and never touch a setting until the run is done.

The topic of "Design of Experiments" (DOE) has great importance in injection molding. Many companies employ this technique, but not effectively. The reason is not because of their lack of knowledge of DOE, but because of their lack of understanding of the basics of molding, along with their choice of factors and levels for the DOE. This topic has been expanded in the new edition.

I would like to thank Hanser Publications and their staff for this opportunity to write the second edition. Mark Smith and Cheryl Hamilton have been very helpful

with the proofing and, moreover, very patient with all the delays from my side. I would also like to thank several other people who have helped me with the second edition. Lorena Castro who took all the bits and pieces of my writing and transformed it into readable flow needs a special mention and acknowledgement.

In the preface to the first edition, I failed to mention a very important place that also helped shape my career and my life. The National Chemical Laboratory (NCL), Pune, India, is where my dad worked all his life as a research scientist. I lived in the shadows of this great institution and its several researchers. My dad would often take me to his lab when he conducted his research, and that is where the seeds of my future were laid. I worked on a couple of projects during my college days in its Polymer Engineering Department, and that was my first personal exposure and involvement with research. It was my experience at NCL, which was one of the contributing factors that pushed me to study further.

My constant sources of inspiration and help include Tim and Violeta of Distinctive Plastics, who have opened their company for my research and seminars, my professor from college, Dr. Basargekar, my colleagues in the industry, Ravi Khare, Atul Khandekar, Vishu Shah, Vikram Bhargava, Randy Phillips, and my family.

To my mom, dad, and siblings, I will be forever indebted to you for all the support and inspiration you have given me over the years.

Suhas Kulkarni
October 2016

1 Introduction to Scientific Processing

1.1 The Evolution and Progress of Injection Molding

Injection molding and extrusion are the most common techniques employed in the manufacture of plastic products. Injection molding of plastics began as an idea by the Hyatt brothers for the manufacture of billiard balls. The idea was borrowed based on a patent by John Smith to inject metal castings. Since then, injection molding of plastics has come a long way. The technique became a popular way to fabricate plastic parts because of the simplicity of the concept, efficiency of production, and the possibility of producing intricate parts with fine details.

The art of injection molding evolved to its present state due to a few key reasons. The requirements of the molded parts became more stringent because of the advances in the fields of science and technology. The demand for tighter tolerances and more complex parts increased and is ever increasing. A required tolerance of a couple thousandths of an inch on a one inch dimension is not uncommon these days. Parts requiring innovative designs, especially designed for assembly (DFA) or parts molded from different materials in the same mold (multi-material molding) are now commonplace. As polymer materials were developed for injection molding, the requirements of processing changed. The discovery of the different morphologies of polymers and the need for better melt homogeneity in molding led to the introduction of the injection screw. Various designs for material-specific screws have followed since. The use of high temperature materials that have high melting points and need high mold temperatures have led to the use of high-temperature ceramic heaters and mold temperature controllers providing higher heat capability. Innovations in electrical and electronic technologies paved the road for machines that could be better controlled, accurate, and efficient. Response times for hydraulic valves can be in milliseconds. All electric machines and hybrid machines are gaining popularity because of their consistency and accuracy. The real time processing parameters of a molding machine can now be viewed from any part of the world via an internet connection and therefore machine production can be monitored or machines can be debugged online. All these features are becoming a common practice among manufacturers. Even some auxiliary equipment can now be debugged and programmed by the suppliers via an internet connection. For the machines tied into the company ERP system, automated messages can be sent to the managers and supervisors about the machine status and quality issues. The need for efficiency and the requirements for advanced product features have dictated the need for innovations in injection molding over the years.

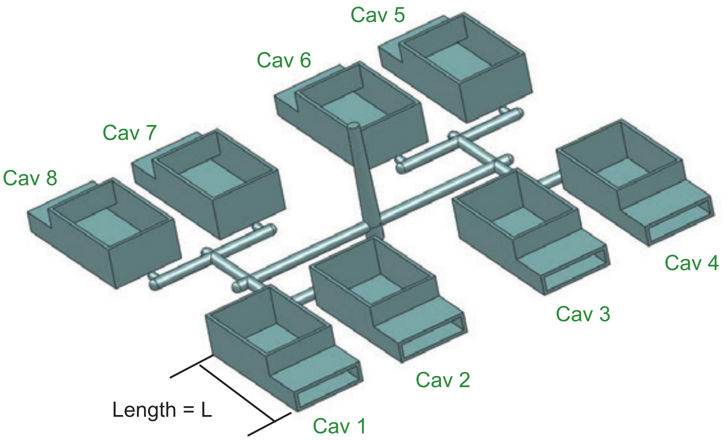
1.2 The Molding Process

The actual molding process has been traditionally defined as the inputs to the molding machine. These are the settings of speeds, pressures, temperatures and times such as injection speeds, holding pressure, melt temperature and cooling time. These are inputs one would set at the molding machine and record on a sheet, commonly called the Process Sheet. However, the word process now needs to be redefined as the complete operation that encompasses all the activities the plastic is subjected to inside a molding facility—from when the plastic enters the molding facility as a pellet to when it leaves the facility as a molded part. For example, the storage of the plastic, the control of the drying of the plastic, and the post mold shrinkage of the part can have a significant influence on the quality of the part. During this journey of the pellet, every stage can have a significant effect on the final quality of the part or assembly. Naturally, understanding every stage now becomes imperative if we would like to control the quality of the molded part. Molding a part that meets the quality requirements is not the real challenge. The real challenge is molding parts consistently; cavity to cavity, shot after shot, and from one production run to another meeting all the quality requirements and with the least amount of effort and maximum efficiency.

1.3 The Three Types of Consistencies Required in Injection Molding

The aim of developing a molding process should be to develop robust processes that would not need any process modifications once the processes are set. Process consistency leads to quality consistency, see Figure 1.1. We look for three different types of consistencies: cavity-to-cavity consistency (Figure 1.1(a)), shot-to-shot consistency (Figure 1.1(b)), and run-to-run consistency (Figure 1.1(c)). Cavity-to-cavity consistency is required in multicavity molds so that each cavity is of the same quality level as the other cavities. Shot-to-shot consistency implies that every consecutive shot would be identical to the previous shot, or the first shot is identical to the last shot of the production run with the process parameters remaining the same during the entire production run. When the process parameters from two different runs are identical and they produce the same quality parts, then this is called run-to-run consistency. Robust and stable processes always yield consistent quality parts with one established process.

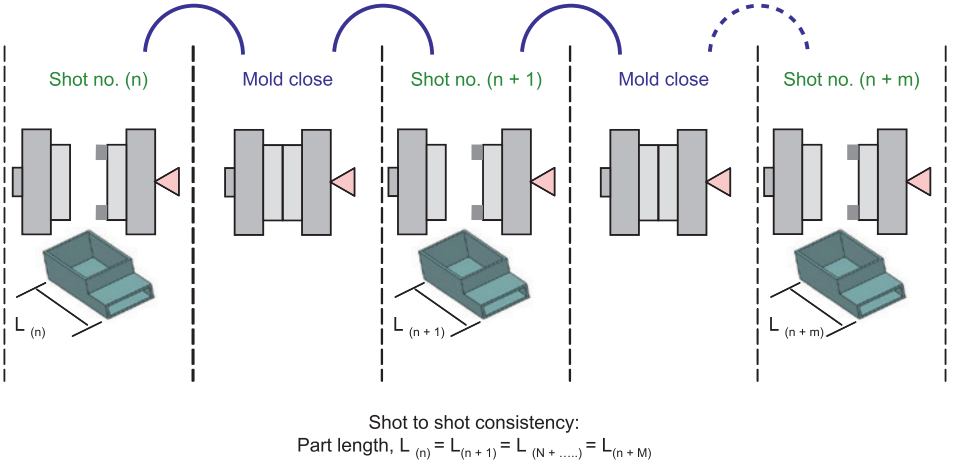
There can be several reasons for the three types of consistencies. A cavity-to-cavity inconsistency could be caused because of an error when cutting the steel in one of the cavities or by making one of the gates too large. A shot-to-shot inconsistency could be caused because of a damaged leaking check ring at the end of the molding screw. A run-to-run inconsistency can be caused because of a lack of a robust process or simply because the process was not accurately or completely documented in the previous run. The run to run consistency is the one that most companies struggle with. This book is deals in depth with process development of robust, repeatable and reproducible processes.



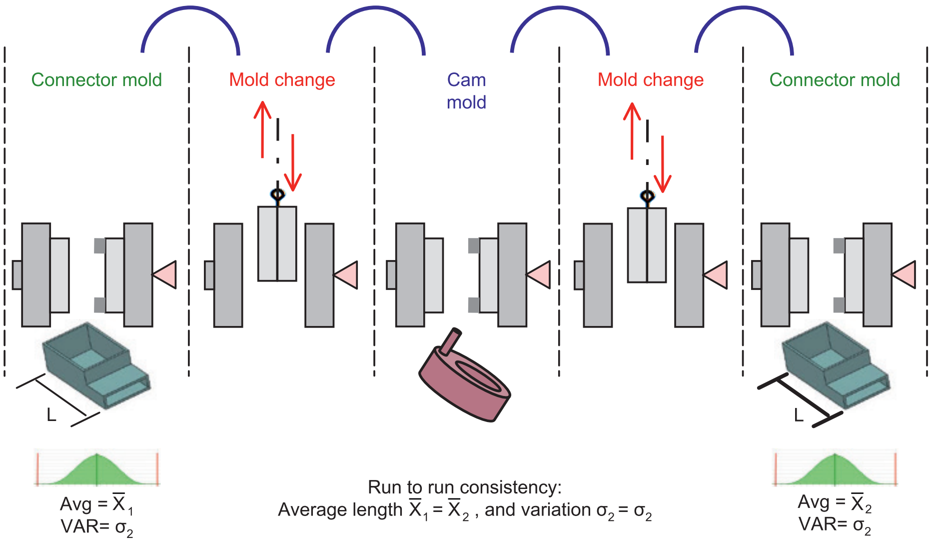
Cavity to cavity consistency:
Part length, $L_{(cav1)} = L_{(cav\ 2)} = L_{(cav\ 3)} = L_{(cav\ 4)}\ L_{(cav\ 5)} = L_{(cav\ 6)} = L_{(cav\ 7)} = L_{(cav\ 8)}$

(a) Cavity to cavity consistency

Figure 1.1 The three types of consistencies required in injection molding



(b) Shot to Shot consistency



(c) Run to run consistency

Figure 1.1 The three types of consistencies required in injection molding *(continued)*

Another reason for inconsistencies and variations in the molded product is the nature of the shrinkage of plastics. When molten plastic is injected inside a mold it cools and freezes to form the product. There is a reduction in the volume of the melt when it cools inside the mold. This is called shrinkage. The magnitude of shrinkage determines the final dimensions of the part. However, this shrinkage is not easily predictable and depends on a number of factors. There is a range of shrinkage values available and that makes it difficult for a mold maker to select a shrinkage value. For example, the shrinkage value for a low density polyethylene is between 1.3 to 3.1 %, which is a wide range. Shrinkage also depends upon the processing conditions. For example, higher the melt temperature, the higher the shrinkage. Almost every processing parameter can affect the shrinkage to varying degrees. Refer to Figure 1.2, which shows the effect of the molding parameter on the length of the part. To increase or decrease the length of the part, several parameters can be increased or decreased.

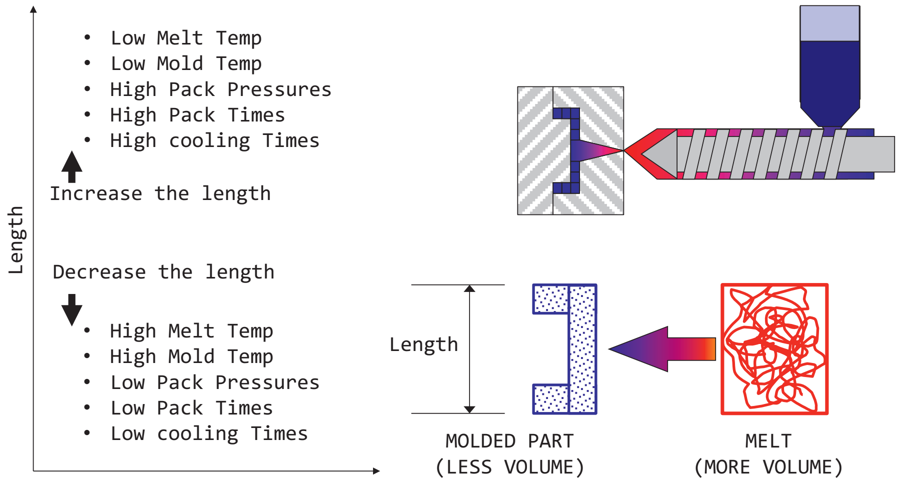


Figure 1.2 Effect of molding parameters on shrinkage and dimension of a part

As seen in the figure, several parameters can have effect on the part dimension and quality. To increase the length of the part, some parameters need to be increased whereas some need to be decreased. Further, the magnitudes of change in length with change in the parameter varies from parameter to parameter. If the molding processes are not developed with these understandings, and in case the dimensions get out of specifications, each processor can work with any one of the parameters. The net result being that processes that were supposedly approved end up having completely different values in a matter of a few runs. When process sheets are compared, for example, from two years ago, there are hardly any numbers that match the current settings.

It should be the goal of every molder to develop an understanding of the molding process for the given mold. A systematic process development approach must be followed. The result of such an approach is a robust, repeatable and reproducible process: the 3 R’s.

A process shown in Figure 1.3 is not acceptable because there is a lot of inefficiency in the system. Such processes result in defective parts, loss of material, loss of time, and not to mention the time and effort put in by the molding personnel. The parts can be remolded and shipped to the customer, however, the time and efforts lost cannot be recovered. The reputation of the molder is something that can also be permanently affected.

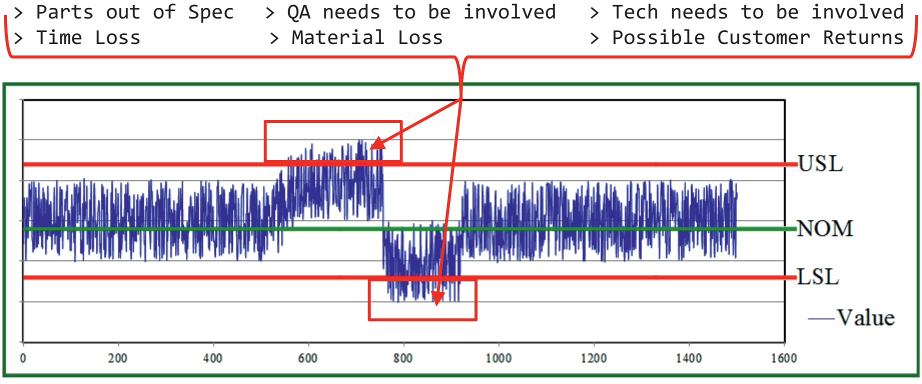


Figure 1.3 Example of an inefficient process

■ 1.4 Scientific Processing

Scientific Processing is the process of achieving consistency in part quality via the application of the underlying scientific principles that control the parameters of the molding process. To achieve this consistency, we must be able to control every activity that is taking place in the process and to control every activity, we must understand the underlying scientific principles. The goal of scientific processing should be to achieve a robust process. Achieving robustness in each of the stages that the pellet travels through automatically translates to an overall robust process. The term consistency must not be confused with the parts being within the required specifications. A consistent process will produce parts that will reflect the consistency but the parts may be out of specifications. In this case, the mold steel must be adjusted to bring the parts within the required specifications and the process must not be altered.

The term *Scientific Molding* was coined and promoted by a two pioneers in the field of injection molding, John Bozzelli and Rod Groleau. Their principles are widely used today and are industry standards. Scientific molding deals with the actual plastic that enters the mold during the molding operation at the molding press.

Scientific processing is the complete process from when the pellet enters the facility and leaves the facility as a finished product. Figure 1.4 shows the journey of the pellet.

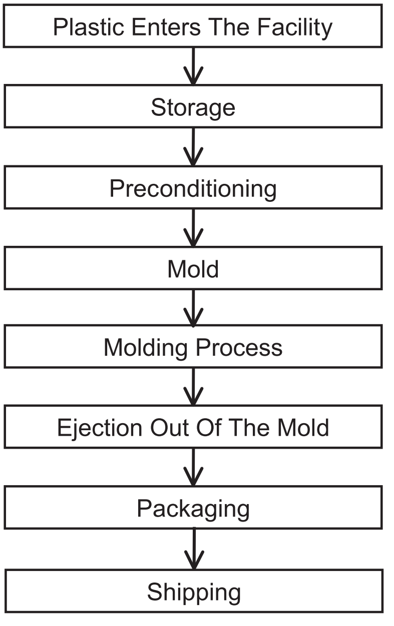


Figure 1.4 The journey of the pellet and the critical factors that need to be controlled

■ 1.5 The Five Critical Factors of Molding

The final molded part is a result of five critical factors, which need to be carefully selected, as shown in Figure 1.5:

1. Part design
2. Material selection
3. Mold design and construction
4. Molding machine
5. Molding process

Each of these factors plays a very important role in the production of the molded part and therefore every one of them has to be optimized for producing the molded part. It is not just the performance of the part but also the consistent molding of the part in production.

1.5.1 Part Design

The concept of the part starts with the engineer designing it. The part must be designed for molding and all the design rules for plastics must be considered. Rules for plastic part design are considerably different than those used for metal part design because of the inherent nature of the plastic. For example, to avoid sink defects in the plastic part, thick sections cannot be present. Additionally, all corners must have a radius to avoid stress concentration and premature failure. With the growing cost of labor and the need for efficiency in the manufacturing process, the part designers now face the added challenges of designing parts for assembly along with those molded parts that utilize multiple materials, commonly referred to as multicomponent molding or multimaterial molding.

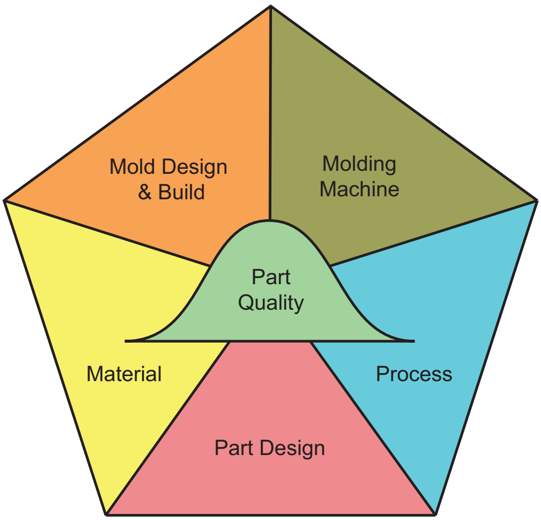


Figure 1.5 The five factors influencing part quality consistency and process robustness

1.5.2 Material Selection

Based on the part design and the part performance requirements, the plastic material must be selected. In addition, the part design may require a special plastic material or a special additive to be added to the base plastic for performance. If a thick section must be present, a filled material may need to be selected or if there is a sliding surface, then an additive reducing the coefficient of friction may need to be added to the plastic. Material selection should typically be done when the basic part design is done. Additional smaller changes can be done concurrently.

1.5.3 Mold Design and Construction

Once the part design and material selection is complete, the mold must be designed and constructed such that it is robust enough to withstand the molding process and the plastic material. For example, during the molding process, the mold can be subjected to high mechanical stresses, especially during the plastic injection and the packing phases. The gates are high-wear areas and there are several places where the air needs to vent out for the plastic to enter the mold. Some plastic materials will require special attention and the mold must be specifically designed with the material in mind. Shrinkage may vary considerably from material to material. All these material specific factors must be considered. The required number of parts over the life of the mold is another factor that will dictate the actual materials of construction. Wear on the mold components must be considered, as the materials chosen to build the injection mold and mold cavities will impact the overall life of the mold and associated amount of maintenance required to keep it production worthy.

1.5.4 Machine Selection

Selecting the right machine for the mold should be done once the mold design is complete. It can be done concurrently during the mold construction stage. The machine plays a very important role in the stability of the molding process. For example, machines with large shot sizes must not be used to mold small shots because the part quality consistency will suffer. Vice versa, using a large percentage of the shot size can give rise to problems with melt homogeneity and therefore issues with fill and dimensions. Small molds must also not be mounted in large machines for fear of mold damage due to excessive clamp tonnage being applied.

1.5.5 Molding Process

Process optimization is the last step before the mold is released into production. This book will cover this topic in detail. If the above four factors and activities are not properly selected or performed, process optimization can be a challenge, if not impossible, without incurring significant cost and delay to the project. At this stage, it is usually very late in the project timeline to make any changes to the part design or mold design, especially because of the cost and time involved. An improperly constructed mold can have a very narrow process window leading to a process that will tend to be unstable. If the material selected is not capable of holding the tolerances, no process will be able to produce satisfactory parts. Molding processes should be robust, repeatable, and reproducable.

1.6 Concurrent Engineering

There are various departments involved in the production of the molded part and therefore regular meetings between the different departments must be held. Each department will have specific knowledge of the selection process and can contribute not just to the process but more importantly predict issues once the mold comes over to their department. For example, getting the process engineer involved in a mold design can help in part orientation in the mold for easy removal, or the mold maker can get help with vent locations based on the process engineer's experience. Involving the quality engineer can help the process engineer understand the required tolerances in the design stage. If the tolerances seem to be unrealistic, they can go back to the product designer for wider tolerances or a material change. There are a lot of benefits associated with implementing concurrent engineering in injection molding. A section is devoted to this topic in this book. In the chapters that follow, the reader will be introduced to the underlying scientific principles to achieve a robust molding process. This understanding will then help in the application of these principles, to develop a robust process and to troubleshoot problems that occur in production. The chapters have been written in a logical sequence to build the readers' knowledge as one would require it or should learn it. However, if the reader is familiar with the topic, he or she can bypass some in favor of other chapters containing the desired information.

1.7 Variation

Variation is a natural phenomenon that is present in every process and activity. For example, the time it takes to drive to work has a number, but it can be an average number that is collected over a certain period of time. There will be times that are lower than the average and there are times that are above the average. In injection molding, if the lengths of 100 parts are measured, then one could get an average number, but there will be parts below and above this number. Variation can never be eliminated, so the goal should be to minimize it. Variation should be measured in order to predict the quality of the molded parts. As shown in Figure 1.6, a molder could measure the part marked as A and decide that all the molded parts are within specification. However, only when the variation is measured can it be seen that there will be some parts, such as the one marked B, which are out of specification.

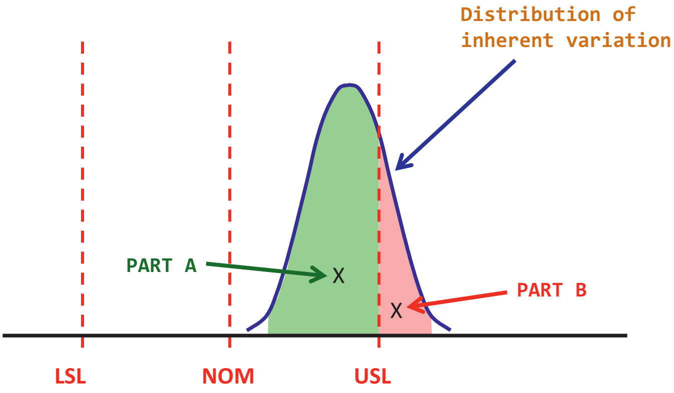


Figure 1.6 Reason to measure variation

Variation in injection molding can come from a number of sources as shown in Figure 1.7. The variation in the molded product is the collective variation from each of these sources, plus many more. Controlling the variation in each source will help the reduction of the overall variation in the final product.

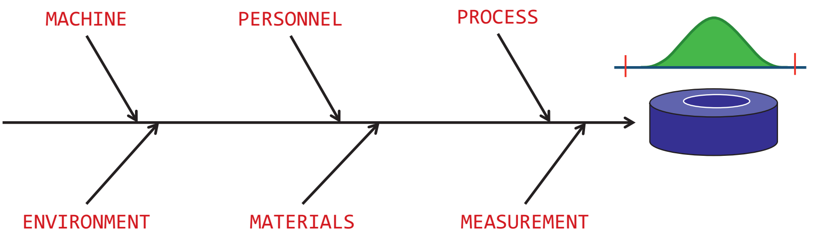


Figure 1.7 Some of the sources of variation in a molded part

Suggested Reading

Osswald, T. A., Turng, L., Gramann, P. J. (Eds.), *Injection Molding Handbook* (2007) Hanser, Munich

Kulkarni, S. M., *Injection Molding Magazine* (June 2008) Cannon Publications, Los Angeles, USA

If such a graph is not available, the machine specification sheet can be helpful. The machine specification sheet should list the maximum plastic pressure. On the molding machine, if the injection pressure setting is selected the machine usually displays a range of pressures. The higher number is the maximum hydraulic pressure of the machine. If the maximum plastic pressure from the specification sheet is divided by the maximum hydraulic pressure of the machine, the resulting value is the IR. In case the machine does not show the range of pressures an input of a very high number in the field will alarm and provide the user with the maximum hydraulic pressure value.

6.9 Selecting the Right Machine for the Mold

The machine selection is one of the five important factors that will contribute to the quality of the part. The mold and the machine must be compatible, and this is often overlooked. Usually, only two factors are taken into consideration: whether the mold physically fits in the machine and whether the clamp tonnage is sufficient. However, one of the most important factors is what percentage of the machine shot size will be used. The residence time of the material in the barrel and the lower limiting size of the mold are other factors. Both are described in the following.

6.9.1 Physical Size of the Mold

The mold size is defined by three variables (see Figure 6.10).

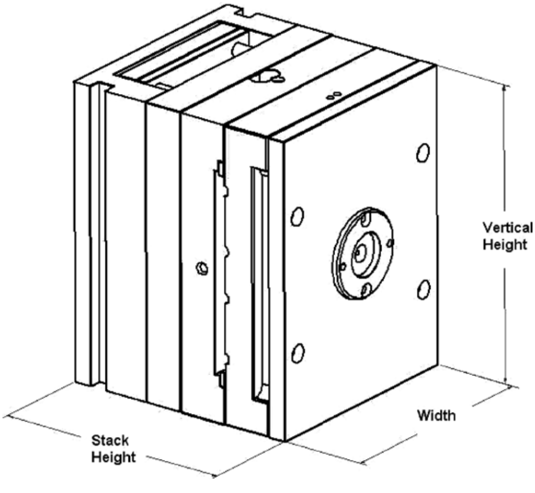


Figure 6.10 Mold size specifications

Mold Stack Height (H): This is the distance between the sides of the mold in the direction of the mold open and close when the mold is fully shut.

Mold Width (W): This is the distance between the vertical sides of the mold, looking in the direction of injection of the plastic. This applies to molds mounted in horizontal machines, but the definitions are extended to other molds also.

Mold Length (L): This is the distance between the top and bottom side of the mold, looking in the direction of injection of the plastic. This again refers to horizontal molds.

Naturally, the mold must fit in the machine such that at least two sides can be bolted to the platen of the machine. The mold can hang off the platens on the other two sides, but the molding area must not be outside the platen. The molding area must always be supported by the platen, see also Figure 6.11. If the cavity is not supported, the injection pressure can easily deflect the plates and cause flash in the part. Injection pressures can be very high, applying tremendous force on the mold base, and over time, can damage the mold components if the mold is not properly supported by the mold platens. On the other hand, the mold must not be considerably smaller than the platen. It must cover at least 70 to 75 % of the area between the tie bars. This is especially true for toggle machines, where the clamping force is applied on the outside and not in the center of the platens. There is a possibility of platen deflection, if the mold is too small, causing platen damage over time. The toggle system also provides reduced support in the center of the mold where the main injection pressure is applied. Even with adequate support pillars in the mold, there still could be deflection because of the lack of support, causing part defects.

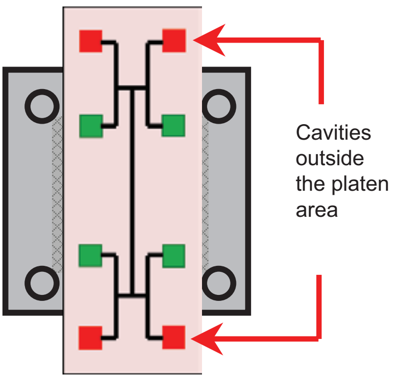


Figure 6.11 Cavities outside the molding area

In regards to the mold height, every machine has a minimum and a maximum mold height that it can accept. The moving platen closes the mold and applies the set tonnage to the mold. Because of the limit on the travel distance during closing, the mold height needs to be greater than this limit. If the mold is less than the minimum mold height, the platen can never let the two halves of the mold touch and apply the tonnage. Therefore minimum mold height is important. On the other hand, the mold must be smaller than the maximum mold height in order to fit in the machine.

The required mold open stroke will depend on the part, usually the part dimension in the direction of ejection. The open stroke should be such that the mold halves are far enough apart when fully opened so that the part falls out of the mold after ejection. The mold open stroke must be greater than the longest dimension of the part in the direction of ejection. For example, on a rectangular part as shown in Figure 6.12, it should be the diagonal of the part. Even with this distance, there is still a danger of damaging the part because it may still hit the side of the mold as it falls off the ejectors. For this reason, the mold opening stroke must be set as wide as possible to avoid damage, but not such that cycle time is lost due to unnecessary movement of the mold.

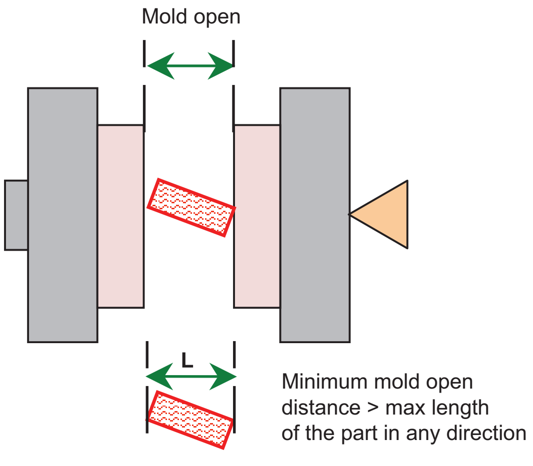


Figure 6.12 Minimum mold open distance

6.9.2 Calculating the Required Machine Tonnage for a Mold

The injection pressure of the plastic applies an outward force on the mold cavities, which works to separate the mold halves. This force must be counter balanced by the machine in order to keep the mold halves closed. If the plastic pressure is higher than the clamp force applied to keep the mold closed, the mold will open, and the plastic will escape from the mold at the parting line where the mold splits open, causing part defects, typically flash. The force that keeps the mold closed is called the clamp tonnage of the machine. The applied clamp force is measured in tons.

The area of the mold that experiences the plastic injection pressure is perpendicular to the direction of mold open and close and is called the projected area. Refer to Figure 6.13. The shaded area is the projected area. In some molds, slides are used to create some of the features of the part. The slides move in and out as the mold closes and opens because of an angled pin or a heel block. The plastic injection pressure can exert a force on the slide and can in turn force the mold open action. In such case, the area of the slide exposed to the plastic must be added on to the projected area.

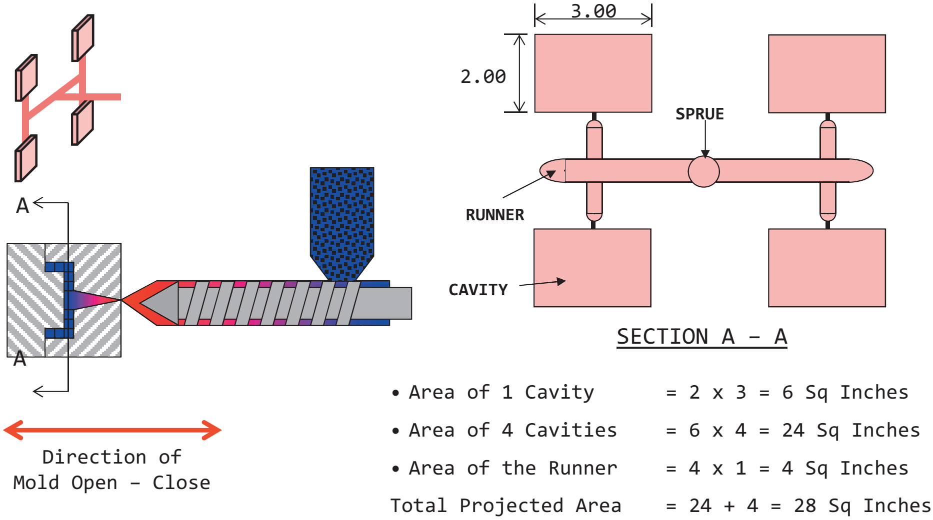


Figure 6.13 Projected area of a mold

The rule of thumb for calculating the tonnage required for the part is given in:

Required tonnage = (Projected area of the part × Number of cavities + Projected area of the runner) × (Tons / in^2 required for the resin)

Depending on the plastic material properties, every material requires a certain amount of force during the mold fill and then requires a certain amount of pressure to pack the part out. Typically, crystalline materials require 3.5 to 4.5 tons of clamp force per square inch of projected area, while amorphous materials require anywhere between 2.5 to 4.0 tons of clamp tonnage per square inch of projected area. The calculation is only a rule of thumb since there are several factors that need to be considered.

■ 6.10 The Rule of Thumb for Tonnage Is Only an Estimate

The calculation mentioned in the previous section is a good estimation but by no means a perfect calculation. Following are the factors that affect tonnage (Refer to Figure 6.14):

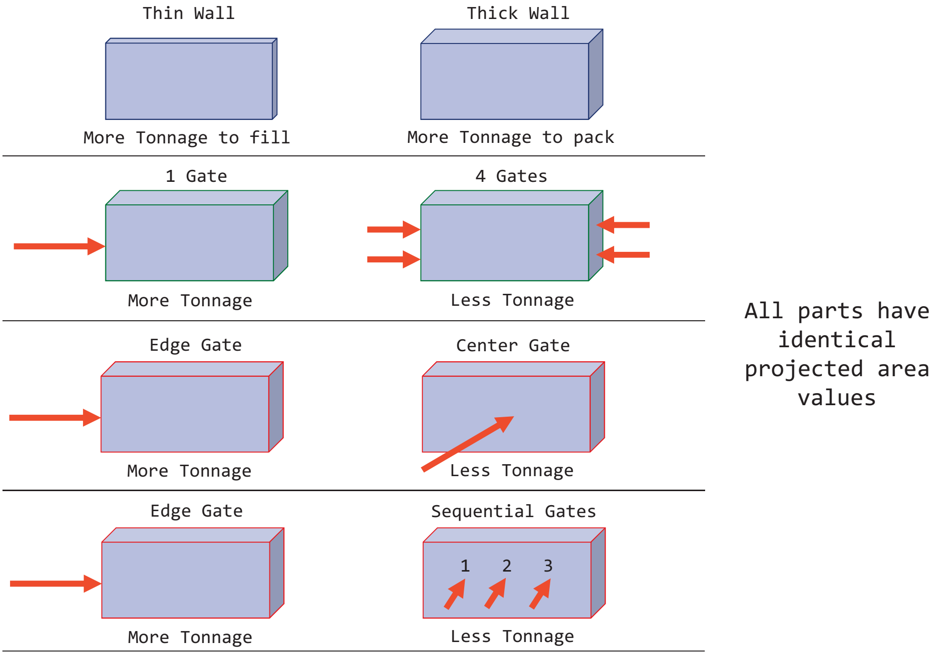


Figure 6.14 Factors affecting required tonnage on parts with identical projected areas

1. Wall thickness: Thinner parts need more pressure to fill the cavity whereas thicker parts will require more packing pressure to compensate for the shrinkage. Two parts can have the same projected area but the thicker part will require more tonnage because it needs to get packed out more than the thinner part. However, in a part such as in a laptop cover, a very thin wall with a long flow length will also require more tonnage to withstand the high injection pressures required to fill the part. Thin walls constitute parts as thin as 0.5 mm (0.020 in) and thick walls are those above 7 to 8 mm (about 0.3 in). Nominal walls are usually between 2 and 5 mm (0.080 to 0.200 in) thick.
2. Number of gates: The more the number of gates, the easier it is to fill the mold and less pressure is required to pack the cavities out. Two parts can have the same projected area but the one with more gates will require less tonnage.

3. Position of the gates: If the part edge is gated, it will require more tonnage compared to as if it is gated in the center because the flow length is cut into half when the part is gated in the center.
4. Sequential valve gating: Molds that are sequentially gated require less tonnage because the force is being applied only in the areas that are influenced by the open gates.
5. Orientation of the part in the mold: In Figure 6.15, the same part is shown to have injection points from two different directions. Using the above formula, the required tonnage when the plastic is injected from the side will be lower than when injected from the front. This does not mean that the part can be run on a lower tonnage press. The flow length would then play a role in the tonnage.

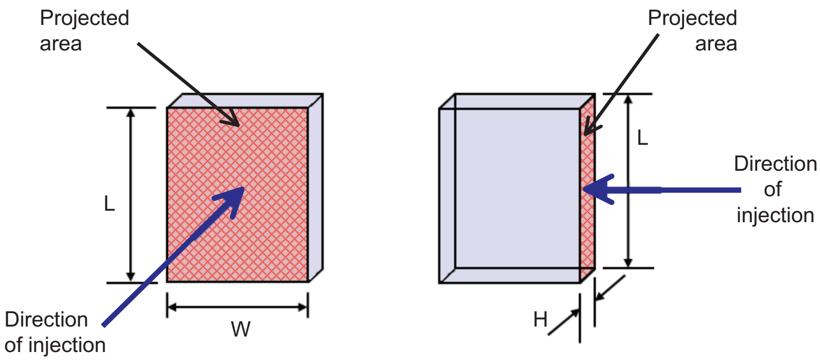


Figure 6.15 Part projected area in the direction of injection

Tonnage calculation is very complex and not easy to predict. Computer simulation programs do an acceptable job in this calculation, but caution is warranted when applying the results.

6.10.1 Percentage Shot Size Used and Number of Shots in the Barrel

The percentage of the shot size used is the most important factor for molding consistency and is often overlooked. The percentage of the shot size gives an idea of the amount of plastic injected into the mold with respect to the maximum amount of plastic the barrel can hold or the fraction of the shot size that is injected into the mold. The formula for this calculation is given in:

% Shot Size used = ((((Part weight × Number of cavities) + Runner weight)) × (1.06/(Density of the plastic)))/(Shot size of the machine) × 100

The percentage of the shot size used must be always between 20 and 80 % of the available shot size.

A number below 20% could result in the following problems:

- The machine needs a finite amount of time to build and achieve the set pressures and speeds. It is similar to listening to a commercial for a car that claim "0 to 60 mph in 5 seconds." This tells us that the car will require 5 seconds to reach 60 mph and so at the end of 2 seconds, the speed is not yet what the driver has set to achieve. Therefore if the molding shot is very small, the injection phase can be inconsistent because not enough time is given for pressure and velocity to reach required levels. Moreover, the plastic that has now built up the pressure is suddenly stopped, and the momentum is unpredictable, leading to large variations in the fill.
- In case of crystalline materials, the shear heat from the turning screw is an important factor for melting the crystallites and achieving a homogeneous melt. If the shot is very small, then the screw may make very few revolutions to reach the set shot size limit. This results in the loss of shear heat and therefore affecting the melt homogeneity.
- With smaller shots, the residence time of the plastic in the barrel increases. This can lead to material degradation.
- Plastic melt is compressible. When pressure is applied to a small shot, some of the applied pressure is lost in compressing the melt leading to the inconsistency in fill. On larger shots sizes, there is a compression of the melt but the percentage compression is much smaller as compared to a smaller shot.

A number above 80% could result in the following problems:

- As described in Section 6.1, the material needs to spend the required amount of time in order going through the melting and homogenization process. A large shot size will transport the material quickly, and the material will not have a chance to form a homogeneous melt for the injection shot. For example, during the startup of a machine, when the machine is being purged using high screw speed and back pressures, sometimes unmelted pellets can be seen coming out of the nozzle tip. This is because the pellets did not have enough time to melt and homogenize because of the higher percentage shot sizes used in purging (Figure 6.16). In the case of hot runner molds, there is an added challenge of transferring the pressure applied to the screw through the hot runner. When the shot is large, the pressure applied has to first compress the plastic in the barrel, then in the hot manifold, and then inject the plastic in the mold. The larger the shot, the more inconsistency.

Figure 6.16 Unmelted plastic pellets in the purge as a result of moving too fast through the molding barrel

Deviation from the 20 to 80% rule:
It is not always possible to have a machine that fits the 20 to 80 % rule for the shot size of a given mold. Almost every molder has such a mold and they can mold perfectly acceptable parts.

On the low end, the preferred number is 20 %. Following are the situations where numbers less than 20 % could be acceptable:

- If the molded part has wide dimensional tolerances. In such cases, any variations from the inconsistent fill that cause a variation in the dimensions can be absorbed by the tolerances.
- If the molded part is being molded out of a heat stable material, such as a polyethylene or a polypropylene. Chances of material degradation is low.
- Recent advances in machinery controls have improved the accuracy and control of the injection phases of the molding cycle. In such cases, deviations from the rule can be considered.

On the high end, the preferred number is 80 %. Unless the screw design is a specially optimized screw for efficient melting with large shot sizes, the rule should never be deviated from. Larger L/D ratio screws help in efficient melting.

As the term suggests, the number of shots in the barrel would mean how many shots are present in the barrel. As an example, if the machine shot size is 60 grams and the injected shot weight is 5 grams, then there would be 4 shots in the barrel. The barrel usage would be 25 %.

Cautionary Notes: Hot runner manufacturers can provide information on the volume of the hot manifold. This volume should not be used in the calculation of the percentage of the shot size. The formula mentioned above is a good estimate because to get the exact percentage shot size number, one needs to add on the amount of melt that is on the molding screw, and at the same time, subtract the material that is not being used for the shot. Only the amount of plastic required for the shot is melted and collected in front of the screw. The machine does not collect the complete maximum shot and then inject what is required.

7 Scientific Processing, Scientific Molding, and Molding Parameters

7.1 Introduction

Several parameters determine a successful molding process. There are various speeds, pressures, times, and temperatures to be considered. Scientific processing encompasses an understanding of the underlying scientific principles of each parameter and the application of these principles to achieve a robust process and consistency in part quality. Scientific processing covers the complete molding process, from the time the plastic enters the facility to when it leaves as a finished product. A robust process is one that can accept reasonable natural variations or a small purposeful change in an input but still delivers consistent output. The term consistency means molding parts with the least variation in the quality of the part. The quality of the part can mean its dimensions, appearance, part weight, or any other aspect that is important to the form, fit, or function of the part. The variation should be from special cause variations and not from any natural cause variations. Special cause variations are variations that are caused by an external factor. For example, if the chiller unit shuts down, the mold temperature will change causing a change in the quality of the part. Natural cause variations are inherent to the process. Their effect can be minimized but not eliminated. For example, if the plastic used to mold the parts has 30% of glass fiber mixed in it, in every molded shot the amount of glass will not be exactly 30%. It will be slightly more or less, for example, between 29.7 and 30.3%. If one weighs 100 consecutive parts from the molding process, each part will weigh differently although the process was not changed. This variation cannot be eliminated, but the mixing process can be improved, and the variation can be reduced.

Robustness and consistency should not be confused with parts being molded within the required specifications. Parts can be out of specifications but the process can be robust and the quality can be consistent. The goal of scientific processing is to achieve a robust process at each stage of the molding process the pellet is subjected to.

The term scientific molding was coined by a two pioneers in the field of injection molding, John Bozzelli and Rod Groleau. Their principles and procedures are widely used today and are industry standards. Scientific molding deals with the actual plastic that enters the mold during the molding operation at the molding machine. The term introduced here is scientific processing, which is defined as the complete activity the plastic is subjected to from the storage of the plastic as pellets to the shipping of the plastic as molded parts. Scientific processing is applying scientific principles to each of the steps involved in the conversion of the plastic to the final product, see Figure 7.1. Chapters 8 and 9 will focus on the understanding and the application of the theories to each of these steps and then optimizing them. Successful process development results in a process that is robust, repeatable, and reproducible.

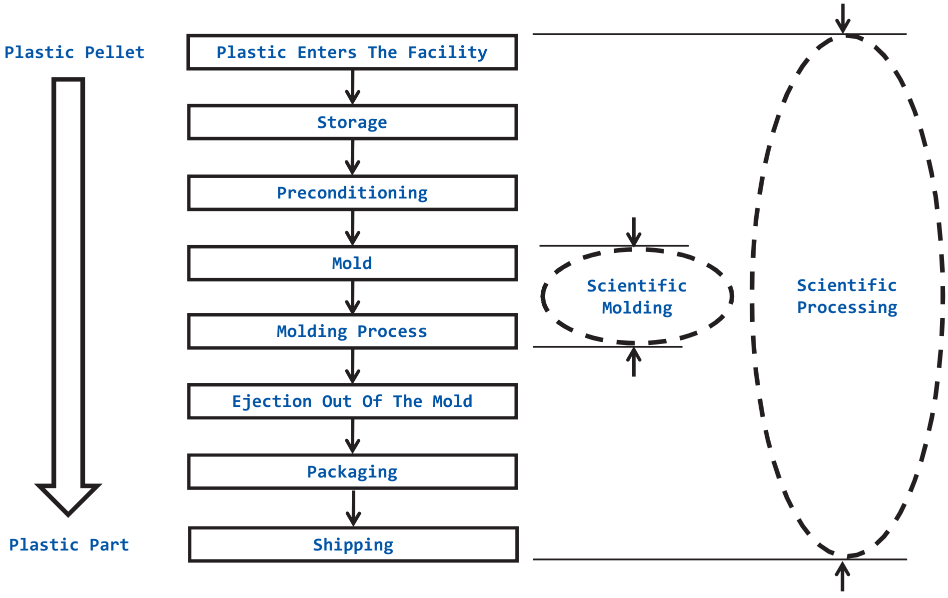


Figure 7.1 The journey of the plastic pellet and the critical factors that need to be controlled

7.1.1 Process Robustness

A process is considered robust when changes to the inputs have minimum effects on the quality of the part. The changes here can be intentional or may be due to natural variations. Naturally, intentional changes must be within reason. In general, a process becomes more robust as larger input changes can be introduced without adversely affecting the resulting output part quality. For example, after a certain injection speed is reached, the viscosity of the plastic remains constant. The viscosity curve is in a robust area and variations in injection speed have little effect on the viscosity and therefore the amount of fill into the mold. At low injection speeds, a slight change in the injection speed causes a large change in the viscosity, resulting in shot-to-shot fill inconsistency. Therefore, this is not a robust area of the process and should be avoided. In addition, it must be understood that natural variations can never be eliminated. Taking these conditions into consideration will help ensure building a robust and consistent process.

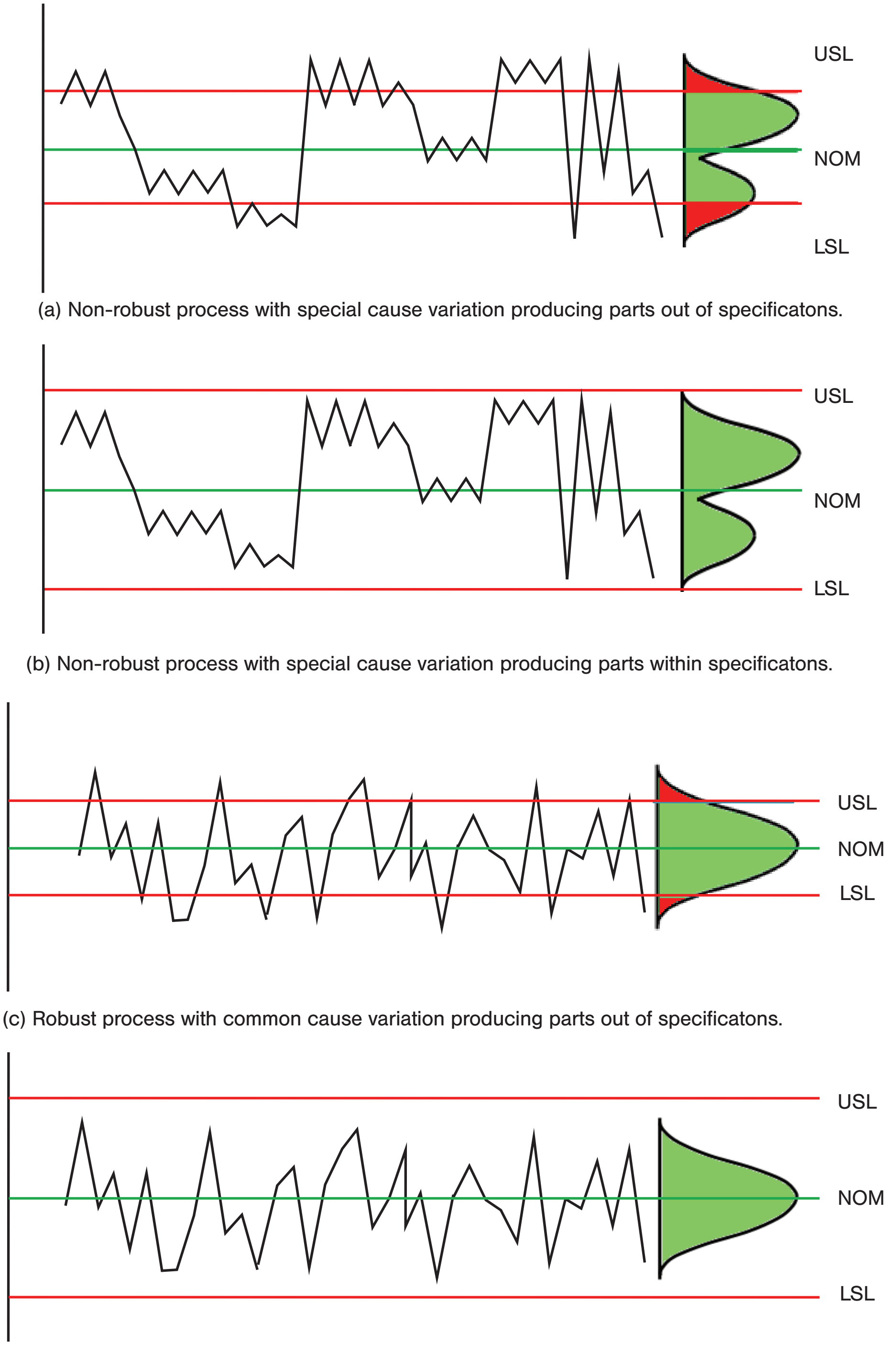
7.1.2 Process Consistency

A process is considered consistent when it meets the following two requirements:

1. All variations in the outputs of the process are a result of only natural cause variation.
2. The standard deviation of the variation is at a minimum value.

For example, the cushion value is an output of the injection, pack, and hold phases. If the cushion value shows minimum variation, and a distribution curve of the cushion value over time is normal, then the process is consistent. In this case, the process under consideration would include only the injection, pack, and hold phases.

A robust process will always produce parts of consistent quality because there is little variation in the output. It also goes without saying that for the quality to be consistent, the process must be robust. For injection molding, whenever there is an inconsistency in part quality, the robustness of the process is usually suspect because the process is reflected in the part quality. In general, based on how robust the process is and on the required tolerance limits, we consider four possible resulting production process scenarios, as shown in Figure 7.2, which shows a representation of a run chart for a particular dimension.



(a) Non-robust process with special cause variation producing parts out of specificatons.

(b) Non-robust process with special cause variation producing parts within specificatons.

(c) Robust process with common cause variation producing parts out of specificatons.

(d) Robust process with common cause variation producing parts within specificatons.

Figure 7.2 Types of processes based on variations and tolerances

Index

M

N

O

P

Q

R