Injection Molding

David O. Kazmer
Department of Plastics Engineering, University of Massachusetts Lowell, Massachusetts, U.S.A.

INTRODUCTION

Injection molding is a well-known process that is capable of economically producing very complex components with demanding specifications.[1] Many different kinds of products, from compact discs to audio and videocassettes, to cutlery and glassware, to automotive parts, are molded using various types of injection molding. In the first half of this entry, a concise review of the process is provided. Then, prevalent guidelines for the design of plastic parts and injection molds are presented.

Continuing global competition is forcing producers of molded parts and suppliers of molding equipment to improve the molding process. Toward this end, an overview of the current state of the art in injection molding is provided in the second half of this entry. Modern machine and process control system designs are presented that provide improved control over the polymer melt. Then, variants of the injection molding process that have been developed to produce molded parts that are hollow, less dense, very thin, or composed of multiple materials are introduced. Finally, a summary of current technological and economic trends in the industry is provided.

PROCESS DESCRIPTION

Injection molding is a net shape manufacturing process in which a polymer melt is forced into an evacuated mold cavity that cools the polymer melt into a desired shape. As shown in Fig. 1, molding machines typically consist of two halves located on opposing sides of a stationary platen: 1) an injection unit that plasticizes the melt and transfers it into the mold, and 2) a clamping unit that closes the mold during the formation of the part and opens the mold for the ejection of the molded product.[2] The design and the operation of molding machines differ substantially, but, in general, adhere to underlying principles that are discussed next.

During the plastication stage, the polymer melt is typically plasticized from solid granules or pellets through the combined effect of heat conduction from the heated barrel and the internal shear heating caused by molecular deformation with the rotation of an internal screw. Screws in injection molding have many similarities with those in single screw extrusion, e.g., most of them have feed, transition, and metering zones. The primary difference, however, is that the screw reciprocates along its axis during the molding cycle. Specifically, the screw moves away from the mold as it rotates and forms a volume of melt at the front of the barrel. The screw then moves forward without turning to force the polymer melt into the injection mold. As such, reciprocating screws in injection molding usually include a nonreturn or check valve to prevent the polymer melt at the front of the barrel from flowing back into the screw during injection. While the dynamics and efficiency of plastication are well known to be a function of the screw design, material properties, and process conditions,[3] many, if not the majority of, molding processes rely on general purpose screw designs that provide reasonable performance for a wide variety of polymer resins.

During the injection stage, the polymer melt is forced from the barrel of the molding machine and enters the mold. Typically, the heated resin travels down a cooled and/or heated feed system, through one or more gates, and into one or more mold cavities where it will form one or more desired shapes. Control of the injection stage is well known to influence the properties of the molded products such as part weight, dimensions, esthetics, orientation, and others.[4–6] As such, modern molding machines allow velocity profiling to control the volumetric flow rate of the polymer melt into the mold during the injection stage. Molding processes are frequently set up so as to best provide a uniform melt front velocity by utilizing a low volumetric flow rate at the start of the injection stage when the polymer melt is just entering the cavity, then a higher volumetric flow rate as the polymer melt diverges and propagates throughout the mold, and then a lower volumetric flow rate as the polymer melt converges toward the end of the mold cavity.

The packing stage typically commences when the mold is filled or nearly filled with the polymer melt and provides additional resin into the mold cavity as the polymer melt cools and contracts. As the packing stage is also crucial to part quality, modern molding machines allow the melt pressure to be profiled as a function of time. The minimum packing pressure is usually set to avoid excessive volumetric shrinkage and

Encyclopedia of Chemical Processing DOI: 10.1081/E-ECHP-120007819

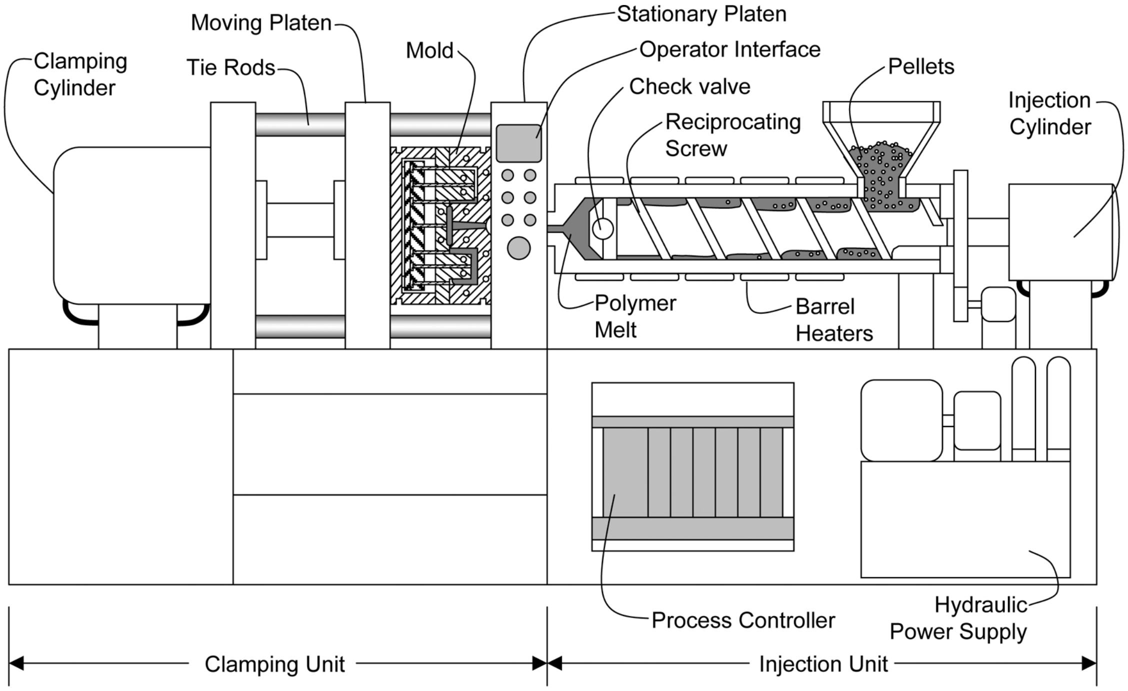

Fig. 1 Schematic of an injection molding machine.

related defects such as sink marks, internal voids, and dimensional instabilities.[7] The maximum packing pressure is usually set to avoid excessive pressure in the mold cavity and related defects such as flashing of the mold, internal stress, and dimensional instabilities.[8] The minimum and maximum packing times are determined by the volumetric shrinkage (required to achieve the desired part dimensions) and the solidification of the gate (which prevents additional polymer melt from entering the cavity at extended pack times), respectively. Accordingly, pack pressure profiling can be used to control the pressure, density, and stress distribution of the polymer melt as it solidifies in the cavity.

After the packing stage, molding processes typically require additional time to plasticate the polymer melt for the next molding cycle while the resin in the cavity solidifies and becomes rigid so that it may be ejected from the mold. Depending on the polymer properties and its inherent recrystallization rate, processing conditions, and the mold design, the cooling stage can range from seconds to minutes.[9] Excessively short cooling times can lead to difficulties in ejecting the part because of either internal compressive stresses that cause the part to remain in the mold cavity or excessive deformation of the part upon actuation of the ejection mechanism(s). Extended cooling times are sometimes used in an attempt to reduce part shrinkage, though this practice can lead to higher internal tensile stresses and difficulties in removing the molded part from the mold core(s) associated with increased ejection forces. Mold release is often utilized to facilitate part removal, though this practice introduces issues related to consistency and contamination. When the polymer melt is appropriately cooled, the molding machine typically actuates the necessary cores, slides, and pins to open the mold and remove the molded part(s).

PRODUCT AND MOLD DESIGN

Injection molding is commonly used because of its ability to form complex parts with features defined in three dimensions, and thereby consolidate the number of components in an assembly, and also reduce the number of fasteners required for the assembly (e.g., Fig. 2). Injection molded parts are usually designed to be thin relative to their length and width dimensions to reduce material and processing costs. As the wall thickness decreases, however, the part becomes very difficult to mold while losing stiffness and strength. As such, part design and mold design are of extreme importance, as a conservative design may result in excessive material and processing costs while an aggressive one may result in inferior performance and require costly design changes.

Computer-aided engineering applications have been developed to optimize the molded part design with respect to manufacturing and end use performance. Many, if not most, commercial molding applications utilize mold filling analyses to analyze the melt flow and pressure distribution in the injection mold before the mold is manufactured. Typically, these analyses

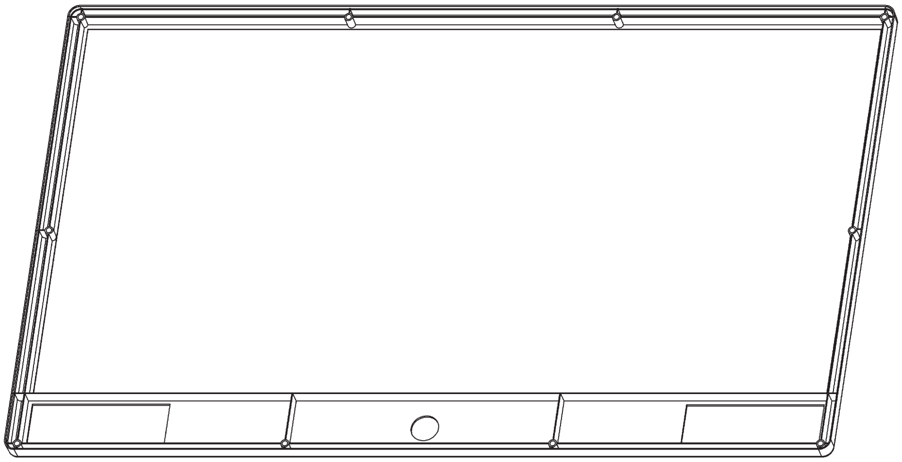

Fig. 2 Front bezel (face plate) for an HDTV with windows, ribs, and bosses.

(e.g., Moldflow, Moldex3D, and Fluent) include non-Newtonian modeling of the polymer viscosity using assumptions for viscous compressible flow. The implemented heat transfer analysis typically includes heat convection with the melt flow, heat conduction from the polymer melt to the metal mold, and internal heat generation from viscous dissipation. In Fig. 3, a typical result for a bezel design in which the flow is introduced at three locations indicated by cones is provided. The contour lines indicate the progressive location of the melt at different times during the mold filling process. For this design, the flow diverges radially from the gates before converging toward the end. This particular design is deficient in that there is a large area of the molded part that has not filled, which would result in localized areas with high melt pressure and nonuniform shrinkage in the molded parts. Most likely, this issue would be rectified by moving the two gates in the window closer together to reduce the flow length.

Computer-aided engineering applications (e.g., Ansys, Cosmos, and Abaqus) are also often utilized to analyze the stiffness and strength of candidate designs. Such concerns often arise in plastic part designs, as 1) injection molded parts are thin relative to their length and width dimensions, and 2) polymeric materials have low elastic modulus and yield stress compared with metals. Because of this, injection molded parts are frequently designed with vertical ribs to ensure adequate stiffness and strength during end use.[10] As a result, molded plastic parts often have stiffness-to-weight ratios that are preferable to alternatively designed metal components.

Because of the widespread use of injection molded components, many guidelines have been developed with respect to the design of injection molding.[11] In general, it is desirable to use a uniform and optimal wall thickness throughout the part to reduce cooling times, and minimize residual stresses and part distortion upon ejection from the mold. Ribs should be used

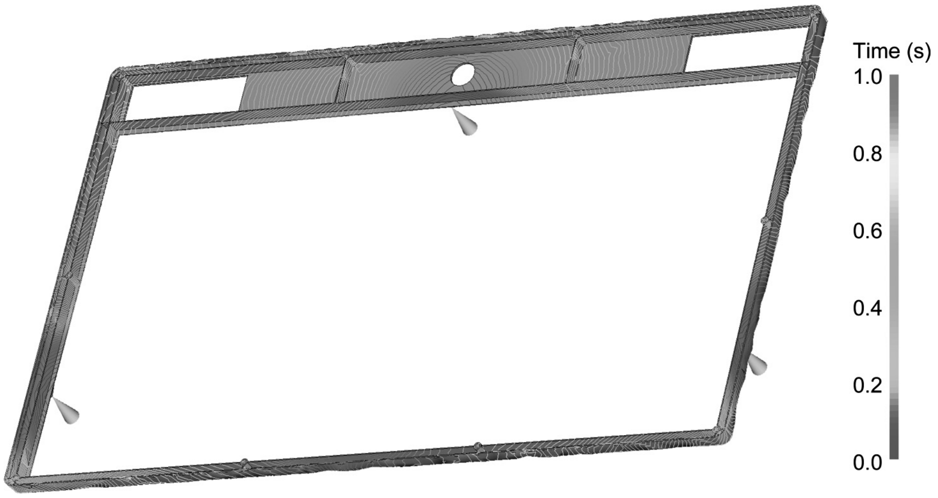

Fig. 3 Progression of polymer melt front into mold cavity. *(View this art in color at www.dekker.com.)*

to improve the overall part stiffness, while gussets to secure bosses and other standing features. Generally, the thickness of the ribs and gussets, and other features should be no more than 70% of the part wall thickness to avoid sink and extended cycle times. It is also desirable to use a generous radius at all corners to avoid flow hesitation and mold hot spots during molding as well as stress concentrations in end use; an inside corner radius equal to the wall thickness is often used. Finally, it is important to design the molded parts to facilitate ejection from the mold by minimizing the number of undercuts that requires core pulls and also providing an acceptable amount of draft in the direction of the part ejection.

The design of injection molds[12] has improved with respect to the level of engineering analysis and manufacturing efficiency. Injection molds (e.g., Fig. 4) must be designed to ensure that the molded parts are economically and consistently produced. For a given molding application, the number of molds and mold cavities is first estimated based on the size and thickness of the part, the location of the gates, and the delivery of the polymer melt to the mold cavity(ies). When possible, the melt pressures are computed using mold filling analysis to estimate the forces likely to be exerted by the polymer melt on the mold components. Structural analysis, in the form of plate bending equations or finite element analysis, may be used to determine the number, location, and size of mold plates and support pillars. Cooling analysis is similarly conducted to determine the number, location, and size of cooling lines in the mold. Based on guidance from the material supplier and prior molding experience, the shrinkage of the plastic is estimated, which is used to determine steel-safe cavity and core designs that will form the molded part(s)?.

Because of the number of subsystems in an injection mold, mold design often requires multiple design iterations. For example, it is not uncommon for the design of the cooling lines to interfere with the placement of ejector pins. As such, the design must be continually refined until the designer is satisfied that the mold will function as desired. As mold making generally requires extensive machining and finishing time, it is quite common for mold makers to order the components and begin rough machining before the detailed mold design is completed. Owing to such concurrent engineering techniques and modern computer controlled machining centers, the cost and time required to produce a high quality mold have decreased significantly over time. After the mold is completed, the mold maker and molder typically perform molding trials to identify and correct any issues prior to the initiation of high volume production.

PROCESS CONTROL

Injection molding machines can control machine elements only with feedback from sensed process data. A fundamental difficulty in polymer processing is that very few of the final part properties can be ascertained within the molding cycle. Instrumentation does not yet exist, and may never exist, to yield information about

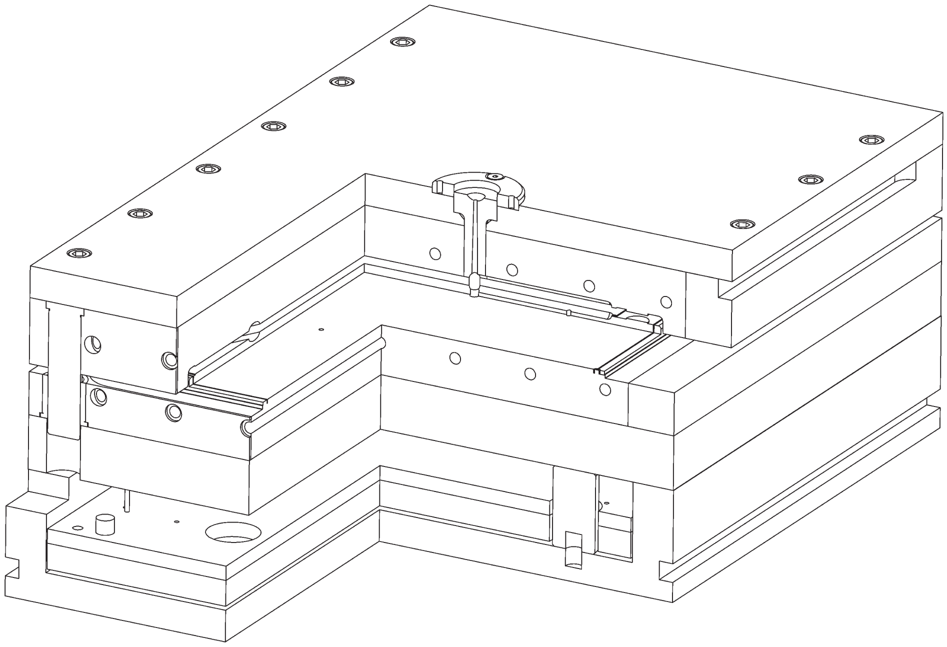

Fig. 4 Partial cutaway section of an injection mold.

esthetics, shrinkage, or structural integrity prior to removal of a plastic part from the mold. As the quality of the molded part is not available, in situ, molding machines operate in an open loop mode with respect to the quality of the molded products. Given the limited number of molding machine control settings, the process capability of injection molding will generally decline with an increasing number of molded parts and related specifications that are simultaneously demanded.[13]

Most machine manufacturers have adopted the use of standard programmable logic controllers in which multiple analog and digital input and output modules are used to acquire or transmit necessary feedback and control signals. The control signals may be generated through the CPU module in the programmable logic controller or alternatively through the use of dedicated axis controllers for faster response. Increasingly, industrial PCs are being used as an interface between the process controller and the operator, such that the PC provides enhanced ease of use, process diagnostics and maintenance, network connectivity, upgradability, and access to third party software.

The dynamics of the molding process are determined through control of different but related machine elements such as motors, heaters, servovalves, etc. These machine elements are typically controlled via a hierarchical closed loop control architecture as shown in Fig. 5.[14] At the innermost level, only the machine elements are regulated by real time comparison of the desired machine set points with the machine feedback, such that the difference (or error) is used to correct the process. At the second level, state variables such as melt temperature and melt pressure are controlled to track prespecified profiles and provide more precise control of the state of the melt. At the outermost level, the machine inputs are adjusted by the machine operator to improve the quality of the part through specification of better set points given feedback of part quality.

The performance of a closed loop controlled molding machine is dependent upon a number of system properties, such as the inertia and dynamic behavior of the process, availability and amount of control energy applied to the machine, the time response and resolution of the sensors providing feedback, and the validity of the control laws that convert perceived errors into corrective actions. Sustained advances in machine design and software algorithms have led to substantial gains in process control performance, such that the molding process control is not usually limited by the sweep time or algorithmic complexity of the controller. While there remains a significant variation in sweep time between molding machines, a control system sweep time (from input of feedback signal to output of control signal) of 2 msec is quite common, with much faster response times widely available. However, the process dynamics are not so dependent upon the response time of the controller, but rather on that of the integrated system. Consider the task of increasing the temperature of a 100 kg steel barrel (heat capacity of 473 J/kg °C) by 10°C, which theoretically requires 473,000 J. If four 1000 W heaters are utilized, then the minimum theoretical response time is about 120 sec or 2 min. Realistically, however, the response time will be much longer than the theoretical minimum, depending upon whether the system is over- or underdamped, and how much error is tolerable. As such, a reduction in the control system response time is likely to be less important than an improvement in the control law, controller tuning, heater design, or barrel design.

While machine control is important, it is the polymer state (pressure, temperature, and morphology) that directly determines the molded part quality. As such, recent technology developments have focused on closing the loop between the machine parameters and the polymer state. Conformal cooling and pulsed cooling are two molding technologies that have been

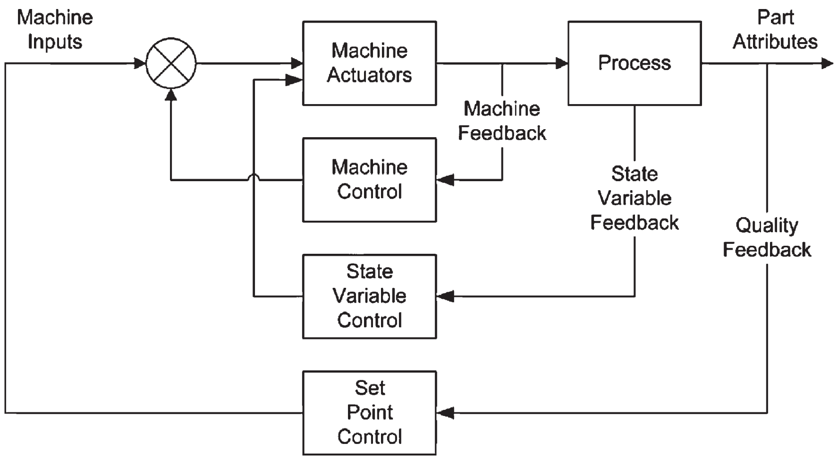

Fig. 5 A three level hierarchical control architecture.

developed to control the state of the melt temperature in the mold cavity. In conformal cooling,[15] the injection mold is manufactured from powdered materials that are deposited in layers and selectively joined with binder from an ink-jet style printhead. Because of this manufacturing process, the mold tooling can be developed with cooling channels designed to be conformal with the molding cavity. Accordingly, such conformal cooling provides the ability to provide a uniform temperature across the molding cavity throughout the process cycle. Pulsed cooling[16] utilizes heaters to affect the surface temperature of the mold. By adding heat prior to and possibly during the molding process, it is possible in the pulsed cooling process to control boundary temperature condition at the polymer–mold interface during the subsequent injection molding cycle. However, cycle time and energy consumption may be increased because of the added heat.

Another fundamental state variable that can be regulated during the cycle is cavity pressure. Closed loop control of cavity pressure could automatically compensate for variations in melt viscosity and injection pressure to achieve a consistent process and consistency of molded products. Adaptive control methods have been developed to track the cavity pressure profile at one location in the mold.[17,18] In these earlier works, cavity pressure control was handicapped by the absence of actuators for distributed pressure control, as conventional molding machines are equipped with only one actuator (the screw), which prevents the simultaneous cavity pressure control at multiple points in the mold. This problem has been solved with the development of dynamic melt flow regulators that allow control of the flow and pressure of the polymer melt at multiple points in the mold.[19,20]

The traditional approach to machine setup in the plastics industry has been trial and error. For this, shots are taken during start-up, and part quality attributes are measured after each shot to evaluate the acceptability of produced parts. The process engineers then use their knowledge of the process to adjust the machine inputs in such a way as to improve the quality of the part from shot to shot. This tuning exercise is repeated until the specifications for part quality are satisfied. The main drawback of the traditional tuning approach is its inefficiency because of its ‘‘ad hoc’’ nature. An alternative to the traditional trial and error approach is the use of expert systems where corrective guidelines are presented in the form of if-then rules.[21] The main shortcoming of expert systems is that a generalized set of rules may not be applicable across a broad range of part geometries, material properties, and machine dynamics.

An analytical alternative for set point specification in more advanced molding operations is to develop an empirical model based on data obtained from a design of experiments (DOE,[22]). Based on the developed empirical models, an objective function of an optimization problem is defined as a function of the part quality attributes. The set of inputs that produces the best set of quality attributes is obtained as the optimal point of this optimization problem. Alternatively, the Extensive Simplex Method has been developed to derive the global feasible process window using a constraint-based approach.[23] Such DOE-based methods offer a systematic approach for process setup,[24] but require significant investment in training and technology. The use of such methods, however, has increased with the increasing interest in Six Sigma and improved process capabilities.

ADVANCED MOLDING PROCESSES

While conventional injection molding is a very capable net shape manufacturing process, process variants are being continuously developed to efficiently produce molded parts with improved properties. In Fig. 6, a classification of several common variants of the injection molding process is provided. As suggested, these variants allow for the molding process to produce parts that are hollow, less dense, very thin, or composed of multiple materials. Other molding technologies have also been developed that are suitable for very high production quantities.

Several molding processes are available for the production of hollow parts. In gas assist[25,26] and water assist[27] molding, the injection mold is designed with thick flow channels. During the molding process, the polymer melt is injected and partially fills the mold. Gas or water is then injected at one or more points and forces the polymer melt to continue flowing down and out of the flow channel and into thinner adjacent sections while the gas or water cores out the molten center of the flow channel. Both processes thereby provide parts with defined external geometry and thicker sections that are coarsely hollowed out (e.g., door handles). Compared with gas assist, water assist provides faster heat transfer and improved internal surface finish, but requires handling of water from the molded part and around the molding machinery.

Injection blow molding and lost core molding are more advanced processes for producing parts with controlled external and internal geometry. In injection blow molding,[28] an injection mold is used to produce a preform with specific thickness and shape. The warm preform is then transferred to a subsequent blowing station where internal pressure is applied and forces the preform to the extremities of a differently shaped mold cavity to provide a hollow part with controlled wall thickness (e.g., soda bottles). In lost core molding,[29] a metal core with low melting temperature is cast with

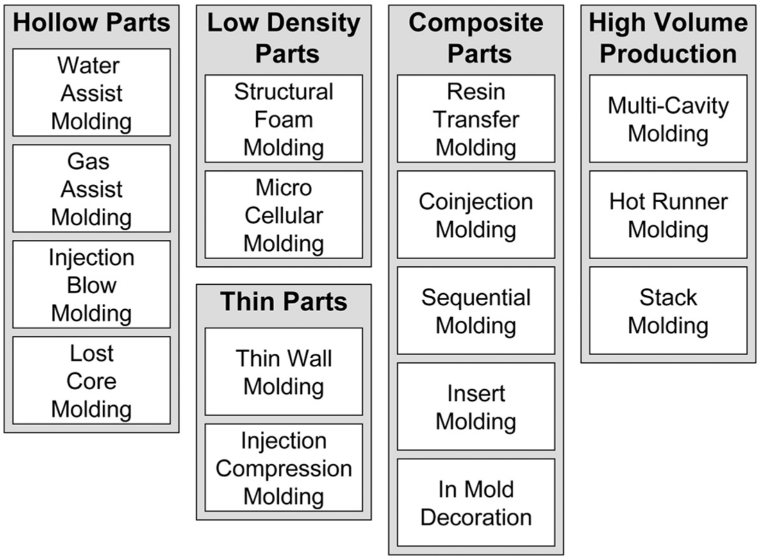

Fig. 6 Classification of various injection molding processes.

complex geometry and placed into an injection mold and then overmolded with the polymer melt per conventional injection molding. After the part is formed and cooled, the core is then melted out, thereby enabling complex internal and external part geometry with repeatable dimensional accuracy (e.g., automotive air intake manifolds) that could not otherwise be made.

Structural foam molding and microcellular molding are processes for creating molded parts with lower density. In structural foam molding[30] (also known as low pressure molding[31]), the polymer melt is used with a reactive blowing agent to create a foamed melt that is injected into the mold cavity. The process is frequently utilized with part designs that are thicker than those produced with conventional injection molding. The resulting molded parts usually exhibit a solid outer skin with an internal foamed core, thus achieving a very high stiffness-to-weight ratio. However, surface finish, foam consistency, and cycle economics are sometimes issues in application. More recently, microcellular molding[32] (also known as MuCell^{TM}) has been developed for application to thinner wall parts. In this process, a supercritical inert fluid is introduced into the polymer melt at a high pressure prior to the injection stage. During injection, the supercritical fluid reduces the melt viscosity, thereby reducing the injection pressure and clamp tonnage. With the subsequent pressure and temperature decay, the supercritical fluid rapidly changes to a gas with very small cell size and low cellular interconnectivity. With adequate process control, the resulting parts can thereby have a significant density reduction without a significant reduction in mechanical properties.

Thin wall molding and injection compression molding are processes utilized to obtain very thin molded parts. In thin wall molding, broadly defined as a mold design in which the length of flow is greater than one or more hundred times its wall thickness, resins with reduced melt viscosity are directly injected at a high pressure into a conventional injection mold.[33] Injection pressures in excess of 200 MPa and sometimes 400 MPa may be utilized to rapidly force the polymer melt throughout the mold cavity, thereby forming thin and complex parts (e.g., laptop housings). Because of the high pressures, thin wall molds are typically designed to be very stiff to avoid deflection during the molding process. In injection compression molding,[34] one side of the mold is displaced during the molding process so as to provide a larger wall thickness during injection of the polymer melt to allow for lower injection pressures. During or after the injection stage, the mold is closed to provide a reduced wall thickness while compensating for volumetric shrinkage. Accordingly, injection compression molding can be utilized to mold thin parts at lower pressures with uniform properties and reduced residual stress (e.g., compact discs[35]).

Given that assembly operations are expensive and can lead to failures, many molding processes have been developed to incorporate multiple materials and/or components into a single composite molding with improved properties and functionality. In resin transfer molding,[36] a composite mat or fabric is placed into a mold. The mold is then closed and the polymer injected into and around the composite fibers. The resulting parts (e.g., automotive body panels) usually have very good structural properties, though surface finish can be an issue. Coinjection molding[37] is another process that utilizes the serial injection of different materials into the mold cavity. Typically, a first material with the desired esthetic and structural properties is injected to form the skin of the molding after which a second material is injected to form the core of the molding. One common application is the molding of fenders and door panels in the automotive industry with a high gloss exterior and internal recycled content.[38] To gain complete control of the distribution of multiple materials in a composite molding, sequential molding[39] (also known as two-shot or multishot molding) is commonly used. In this process, a molded part is produced in a first mold cavity with a first polymeric material. The part is then transferred to a second mold cavity into which a second polymeric material is injected, which fuses with the first molded part. The resulting parts may consist of multiple colors (e.g., automotive tail lights) or structural properties (e.g., soft grip razors).

Insert molding[40] can be considered a more generic version of sequential molding, as most components can be placed into an injection mold and overmolded with a polymeric material. Common applications of insert molding include the integration of electrical connectors in the housings of electrical devices as well as the incorporation of metal stiffeners and fasteners in structural components. Recently, in-mold decoration has become common.[41] In this process, a thin decorative film is inserted into the mold cavity and overmolded with the polymer melt. The resulting parts (e.g., cell phone faceplates) typically have complex printed patterns between the highly glossy surface and the polymer substrate.

While a single injection mold cavity may provide hundreds of thousands of moldings, applications requiring higher production volumes typically require multiple cavities to increase production rates and reduce production costs. In a multiple cavity mold, two or more cavities are connected via a runner system to the nozzle of the molding machine.[42] In a naturally balanced runner system, the mold geometry is symmetric such that the same melt flow and pressure are provided to each mold cavity. In this manner, a number of molded parts can be produced in the time normally taken to produce one molded part. Unfortunately, the runner system can utilize a significant amount of polymeric material and require extended cooling times to adequately solidify for ejection. To avoid excessive waste across many molding cycles and also to enable the operation of a higher number of mold cavities, the mold may be fitted with a hot runner system. In a hot runner system, the polymer melt is transmitted through a series of heated channels directly to the mold cavity. As a result, the polymer melt remains molten in the hot runner system and there is no wasted material or time. Typically, hot runner molding also improves molded part quality because of the improved melt transmission between the nozzle and the mold cavities.[43]

A difficulty with multicavity molding is that the force exerted by the melt onto the molding machine is proportional to the number of mold cavities. To reduce these clamping forces while operating with a higher number of cavities, stack molding was developed.[44] In stack molding, a second set of cavities is placed directly behind a first set of cavities in an injection mold. Both sets of cavities are molded at the same time, with the mold opening on two parting planes for ejection of the parts. As the molding pressures within the sets of mold cavities are opposing, two sets of mold cavities can be operated with the clamping forces required to mold only one set of cavities. As such, stack molds provide for very rapid and economical production of a large quantity of molded parts, given the additional upfront investment in molding technology.

CONCLUSIONS

While injection molding is a widely used and mature process, technological advances in computer-aided engineering and process control have brought new capabilities with respect to product design, mold making, and molding machine design. Product designs have become more complex and lightweight, with an ever-increasing number of designs incorporating composite material systems. Mold making has become extremely efficient in which the mold design utilizes current solid modeling techniques with libraries of mold components, and the mold manufacturing utilizes numerical controlled programming derived from the mold design on numerically controlled machining centers. Molding machines have become more capable and energy efficient with advances in machine components as well as easier to use with the incorporation of computer technology into user interfaces.

The selection adoption of advanced molding technologies is necessary to remain competitive. There has been a long term trend of consolidation by mergers and acquisition in the plastics industry. The resulting larger producers generally have a broader technology portfolio compared with the smaller producers. The smaller producers are responding by focusing on niche applications with a specialized technology strategy. These industry conditions are dynamic, but provide for very competitive services to customers.

ARTICLES OF FURTHER INTEREST

Process Optimization, p. 2439.

REFERENCES

1. Isayev, A.I. *Injection and Compression Molding Fundamentals;* Vol. 1, Marcel Dekker, 1987.
2. Johannaber, F. *Injection Molding Machines: A User's Guide*; Hanser Gardner Publications, 1994.
3. Rauwendaal, C. *Polymer Extrusion*, 4th Ed.; Hanser Gardner Publications, 2001.
4. Mann, J.W. Process parameter control: the key to optimization. Plast. Eng. **1974**, *30*, 25–27.
5. Lord, H.A.; Williams, G. Mold filling studies for the injection molding of thermoplastic materials: transient flow of plastic materials in the cavities of injection molding dies. Can. Controls Instrum. **1975**, 318–328.
6. Pandelidis, I.O.; Agrawal, A.R. Optimal anticipatory control of ram velocity in injection molding. Polym. Eng. Sci. **1988**, *28*, 147–156.
7. Chang, R.Y.; Tsaur, B.D. Experimental and theoretical studies of shrinkage, warpage, and sink marks of crystalline polymer injection molded parts. Polym. Eng. Sci. **1995**, *35*, 1222.
8. Zheng, R.; Kennedy, P.; Phan-Thien, N.; Fan, X.J. Thermoviscoelastic simulation of thermally and pressure induced stresses in injection molding. J. Non-Newtonian Fluid Mech. **1999**, *84*, 159–190.
9. Xu, H.; Kazmer, D.O. A stiffness criterion for cooling time estimation. Int. Polym. Process. **1999**, *13*, 249–255.
10. Trantina, G.; Nimmer, R. *Structural Analysis of Thermoplastic Components*; Hanster Gardner Publications, 1994.
11. Malloy, R. *Plastic Part Design for Injection Molding: An Introdution*; Hanser Gardner Publications, 1994.
12. Menges, G.; Michaeli, W.; Mohren, P. *How to Make Injection Molds*, 3rd Ed.; Hanser, 2001.
13. Kazmer, D.; Roser, C. Evaluation of product and process design robustness. Res. Eng. Design **1999**, *11*, 20–30.
14. Wang, K.K. Intelligent process control using on-line models. NSF Design and Manufacturing Grantees Conference, Los Angeles, CA: 1999.
15. Sachs, E.; Wylonis, E.; Cima, M.; Allen, S.; Michaels, S.; Sun, E.; Tang, H.; Guo, H. Injection molding tooling by three dimensional printing, a desktop manufacturing process. Proceedings of the society of plastics Engineers Annual Technical Conference, ANTEC, Society of Plastics Engineers: Brookfield, CT, 1995; Vol. 1, 997–1003.
16. Jansen, K.M.B.; Flaman, A.A.M. Influence of surface heating on the birefringence distribution in injection molded parts. Polym. Eng. Sci. **1994**, *34*, 898–904.
17. Gao, F.; Patterson, I.A.N.; Kamal, M.R. Self-tuning cavity pressure control of injection molding filling. Adv. Polym. Technol. **1994**, *13*, 111–120.
18. Nunn, R.E.; Grolman, C.P. Closed loop cavity pressure control in injection molding. J. Reinforced Plastics Composites **1991**, *9*, 2121.
19. Kazmer, D.O.; Barkan, P. Multi-cavity pressure control in the filling and packing stages of the injection molding process. Polym. Eng. Sci. **1997**, *37*, 1865–1879.
20. Kazmer, D.; Kudchakar, V.; Nageri, R. Performance of a self-regualating melt pressure valve. Proceedings of SPE Annual Technical Conference; 2005.
21. Shelesh-Nezhad, K.; Siores, E. Intelligent system for plastic injection molding process design. J. Mater. Process. Technol. **1997**, *63*, 458–462.
22. Myers, R.H.; Montgomery, D.C. Response surface methodology: process and product optimization using designed experiments. *Wiley Series in Probability and Statistics*; Wiley Interscience, 1995; 248 pp.
23. Zhu, L.; Kazmer, D. An extensive simplex method mapping the global feasibility. J. Eng. Optimization **2003**, *35*, 165–176.
24. Liu, C.; Manzione, L.T. Process studies in precision injection molding. I. Process parameters and precision. Polym. Eng. Sci. **1996**, *36*, 1–9.
25. Rothe, J. Improvements to injection moulding machinery. Kunststoffe German Plastics **1988**, *78*, 19–23.
26. Turng, L.S. Development and application of CAE technology for the gas-assisted injection molding process. Adv. Polym. Technol. **1995**, *14*, 1–13.
27. Liu, S.-J.; Chen, Y.-S. Water-assisted injection molding of thermoplastic materials: effects of processing parameters. Polym. Eng. Sci. **2003**, *43*, 1806–1817.
28. Abramo, R.J. Injection blow molding PVC containers. J. Vinyl Technol. **1992**, *14*, 87–92.
29. Michaeli, W.; Ziegmann, C. Micro assembly injection moulding for the generation of hybrid microstructures. Microsyst. Technol. **2003**, *9*, 427–430.
30. Wang, S.F.; Ogale, A.A. Structural foam molding. Modeling of bubble-growth process resulting from chemical blowing agents. Polym. Plast. Technol. Eng. **1990**, *29*, 355–369.
31. Shallenberg, D. Low pressure structural foam molding carry the load with style. Presented at Society of Manufacturing Engineers Annual Meeting, 1999; Vol. CM99–193, 1–13.
32. Yoon, J.D.; Hong, S.K.; Kim, J.H.; Cha, S.W. A mold surface treatment for improving surface

finish of injection molded microcellular parts. Cell. Polym. **2004**, *23*, 39–47.

33. Fassett, J. Thin wall molding: how its processing considerations differ from standard injection molding. Plast. Eng. **1995**, *51*, 35–37.
34. Yang, S.Y.; Nien, L. Experimental study on injection compression molding of cylindrical parts. Adv. Polym. Technol. **1996**, *15*, 205–213.
35. Fan, B.; Kazmer, D.O.; Bushko, W.C.; Theriault, R.P.; Poslinski, A.J. Simulation of injection-compression molding for optical media. Polym. Eng. Sci. **2003**, *43*, 596–606.
36. Lee, C.-L.; Wei, K.-H. Resin transfer molding (RTM) process of a high performance epoxy resin. Polym. Eng. Sci. **2000**, *40*, 935–943.
37. McRoskey, J. Coinjection advances open up new applications. Plast. World **1984**, *42* (9), 50–51.
38. Moss, M. Sequential coinjection hot runner. Proceedings of the society of Plastics Engineers Annual Technical Conference. ANTEC, Society of Plastics Engineers: Brookfield, CT, 1998; Vol. 1, 351–357.
39. Wilder, R.V. Multicomponent molding. Now a strategic capability Modern Plast. **1990**, *67*, 65–69.
40. Strasser, F. Molding metal inserts into plastic parts. Plast. Eng. **1980**, *36*, 17–22.
41. Enewoldsen, P.; Braun, H. In-mould decoration. Kunststoffe Plast Europe **1999**, *89*, 35–36.
42. Beaumont, J. *Runner and Gating Design Handbook: Tools for Successful Injection Molding*; Hanser, 2004.
43. Ohnuma, S. 64-Impression hotrunner mould for TPE cuff seals. Kunststoffe German Plastics **1989**, *79*, 17.
44. Strauch, R. Development and construction of a stack mold. Kunststoffe German Plastics **1988**, *78*, 15–16.