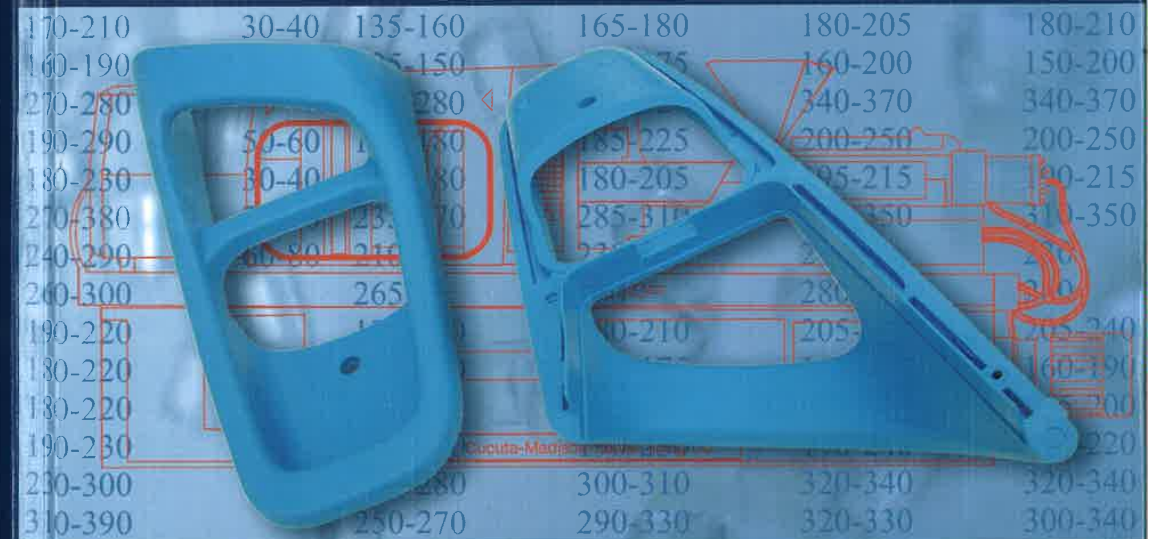# Injection Molding Handbook

Edited by
Osswald / Turng / Gramann

HANSER

Injection Molding Handbook

Edited by

Tim A. Osswald,
Lih-Sheng (Tom) Turng,
and
Paul J. Gramann

with contributions from

J. Beaumont, J. Bozzelli, N. Castaño, B. Davis, M. De Greiff, R. Farrell,
P. Gramann, G. Holden, R. Lee, T. Osswald, C. Rauwendaal, A. Rios,
M. Sepe, T. Springett, L. Turng, R. Vadlamudi, J. Wickmann

HANSER

Hanser Publishers, Munich
Hanser Gardner Publications, Inc., Cincinnati

Tim A. Osswald, Department of Mechanical Engineering, Polymer Engeneering Center, Madison, WI 53706, USA
Lih-Sheng (Tom) Turng, Department of Mechanical Engineering, Polymer Engeneering Center, Madison, WI 53706, USA
Paul J. Gramann, The Madison Group: PPRC, Madison, WI 53719, USA

We dedicate this handbook to Professor Kuo-King (K. K.) Wang whose vision and pioneering contributions propelled the advancement of injection molding technology.

Tim A. Osswald, Paul J. Gramann, and Lih-Sheng (Tom) Turng

molding processes is to use x and Rm charts (individual measurement and moving range) for each individual stream. This method is recommended for cases where the CpK of the cavities is equal to or greater than 3 ($CpK \geq 3$). When $CpK < 3$, the x-bar and R chart (average and range) is recommended. This approach, however, tends to be complicated and time consuming.

An alternative and more expedient approach is to use the group chart. In this chart the highest and lowest values are plotted on the x-chart and the largest moving range on the R-chart. An example of a group chart is shown in Fig. 8.30.

The process is running well as long as the high x-value is below the upper action limit and the low x-value above the lower action limit. The moving range chart shows the maximum range from any stream to its own previous value, making it sensitive to changes in any stream regardless of constant differences between streams. The group chart is most useful for day-to-day monitoring; however, a tabular report can be useful to identify consistently high or low stream. For instance, from a tabular report it may become obvious that cavity 3 is running consistently high. If we use an M/I chart, we can denote the individual points with the cavity number. In this case, it will be immediately obvious from the chart when one of the cavities is running consistently low or high. Identifying the low and high X values with the cavity number will do the same in the group chart. This method is preferred because it is easier to identify problems from a chart than from tabular data.

References

1. Shewhart, W., *Economic Control of Quality of Product* (1931), Van Nostrand Reinhold, New York.
2. Grant, E. I., Leavenworth, R. S., *Statistical Process Control*, 5th ed. (1980), McGraw-Hill, New York.
3. van der Veen, J., Holst, P., *Median/Individual Measurements Control Charting and Analysis for Family Processes* (1993), Northwest Analytical, Inc.
4. Bajaria, H., Skog, F., *Quality* (1994), December.
5. Rauwendaal, C.J., Statistical Process Control in Injection Molding and Extrusion (2000), Hanser, Munich.

9 Special Injection Molding Processes

L.-S. Turng*

Injection molding is one of the most versatile and important operations for mass production of complex plastic parts. The injection molded parts typically have excellent dimensional tolerance and require almost no finishing and/or assembly operations. In addition to thermoplastics and thermosets, the process is also being extended to such materials as fibers, ceramics, and powdered metals, with polymers as binders. Among all the polymer-processing methods, injection molding accounts for 32% by weight of all the polymeric material processed [1]. Nevertheless, new variations and emerging innovations of conventional injection molding have been continuously developed to extend the applicability, capability, flexibility, productivity, and profitability of this process further. To be more specific, these special and emerging injection molding processes introduce additional design freedom, new application areas, unique geometrical features, unprecedented part strength, sustainable economic benefits, improved material properties and part quality, and so on, that cannot be accomplished by the conventional injection molding process.

This chapter is intended to provide readers with a general introduction of these special injection molding processes with emphases on process description, relevant advantages and drawbacks, applicable materials, as well as existing and/or potential applications. References listed at the end of this chapter provide detailed information for more in-depth studies. With this information, readers will be able to evaluate the technical merits and applicability of the relevant processes in order to determine the most suitable production method. It is also hoped that a collective presentation of these various special molding processes, which descend from the same origin and yet mature with diversified creativity, will spark innovative ideas that lead to further improvement or new inventions.

In addition, thanks to a well-focused research effort conducted at many research and educational institutions (see, e.g., [2–20]), a solid scientific foundation for injection molding and related special processes discussed in this chapter has been established. Based on the resulting findings and theoretical principles, computer-aided engineering (CAE) tools have been developed and are now widely used in the industry. As a result, the design and manufacturing of injection-molded parts have literally been transformed from a "black art" to a well-developed technology for many manufacturing industries. These CAE tools help the engineer gain process insight, pinpoint blind spots and the problems usually overlooked, and contribute to the development and acceptance of many special injection molding processes discussed in this chapter.

*With a contribution from Mauricio Degreiff and Nelson Castaño on rubber injection molding.
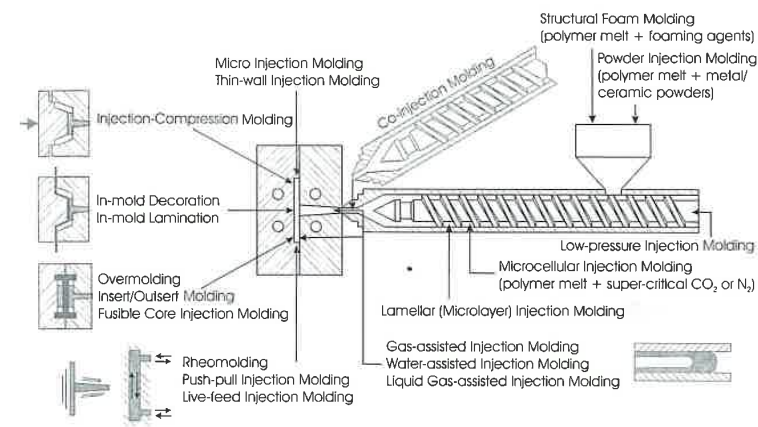
Figure 9.1 Special injection molding processes for thermoplastics.

It is very difficult to cover all special injection molding processes, not to mention those new processes that are being developed and field-tested. Furthermore, due to the diversified nature of these special injection-molding processes, there is no unique method to categorize them. As a preliminary attempt, Table 9.1 classifies the various processes based on the specific techniques employed by the process or the unique characteristics of the process. Figure 9.1 illustrates some of the characteristics of those special injection molding processes for thermoplastics. It should be noted that, for a special purpose or application, a new or viable special injection molding process could employ multiple specific techniques listed in Table 9.1 (e.g., gas-assisted powder injection molding, multicomponent powder injection molding, gas-assisted push-pull injection molding, coinjection molding with microcellular plastics, etc.). Detailed description of these processes can be found in the following sections dedicated to each individual process.

9.1   Coinjection (Sandwich) Molding

Coinjection molding (sometimes called "sandwich molding") comprises sequential and/or concurrent injection of a "skin" material and a dissimilar but compatible "core" material into a cavity. This process produces parts that have a sandwich structure, with the core material embedded between the layers of the skin material. This innovative process offers the inherent flexibility of using the optimal properties of each material to reduce the material cost, injection pressure, clamping tonnage, and residual stresses to modify the property of the molded part, and/or to achieve particular engineering effects.

Table 9.1   Categorization of Special Injection Molding Processes

1. Incorporation of additional material(s) or component(s) into the molded part
   a. Adding or injecting additional plastics
      i. Coinjection molding
      ii. Multi-component injection molding (overmolding)
      iii. Lamellar (microlayer) injection molding
   b. Injection around (or within) metal components
      i. Insert/outsert molding
      ii. Fusible core (lost core) injection molding
   c. Injecting gas into the polymer melt
      i. Gas-assisted injection molding
   d. Injecting liquid or water into polymer melt
      i. Liquid gas-assisted injection molding
      ii. Water-assisted injection molding
   e. Injecting gas into the metal (or ceramic) powder-polymer mixture
      i. Gas-assisted powder injection molding
   f. Incorporating reinforced fiber mats inside the cavity
      i. Resin transfer molding
      ii. Structural reaction injection molding
   g. Incorporating film, foil, fabric, or laminate to be back-molded by polymer melt
      i. In-mold decoration and in-mold lamination
      ii. Low-pressure injection molding
2. Melt formulation
   a. Mixing polymer melt with super-critical fluids
      i. Microcellular injection molding
   b. Mixing polymer melt with chemical or physical blowing agents
      i. Structural foam injection molding
   c. Mixing polymer melt with metal or ceramic powders
      i. Metal/ceramic powder injection molding
   d. Mixing prepolymer (monomers or reactants) prior injection
      i. Reaction injection molding
      ii. Structural reaction injection molding
      iii. Resin transfer molding
      iv. Thermoset injection molding
3. Melt manipulation
   a. Providing vibration and oscillation to the melt during processing
      i. Multi live-feed injection molding
      ii. Push–pull injection molding
      iii. Rheomolding
      iv. Vibration gas(-assisted) injection molding
   b. Using screw speed and back pressure to control melt temperature
      i. Low-pressure injection molding
4. Mold movement
   a. Applying compression with mold closing movement
      i. Injection-compression molding
5. Special part or geometry features
   a. Producing parts with miniature dimensions or relatively thin sections
      i. Micro-injection molding
      ii. Thin-wall molding

9.1.1 Process Description

Coinjection is one of the two-component or multi-component injection molding processes available today (see Table 9.2). Unlike other multi-component molding processes, however, the coinjection molding process is characterized by its ability to encapsulate an inner core material with an outer skin material completely. The process mechanics rely on the sequential and/or concurrent injection of two different materials through the same gate(s). Figure 9.2 illustrates the typical sequences of the co-injection molding process using the "one-channel technique" and the resulting flow of skin and core materials inside the cavity. This is accomplished with the use of a machine that has two separate, individually controllable injection units and a common injection nozzle block with a switching head. The principle of this type of process is relatively simple: Two dissimilar polymer melts from their injection units are injected one after the other into a mold cavity. In particular, in stage (a) of Fig. 9.2, a short shot of skin material (shown in black) is injected into the mold. Due to the flow behavior of the polymer melts and the solidification of skin material, a frozen layer of polymer starts to grow from the colder mold walls. The polymer flowing in the center of the cavity remains molten. As the core material is injected, it flows within the frozen skin layers, pushing the molten skin material at the hot core to the extremities of the cavity, as shown in stage (b) of Fig. 9.2. Because of the fountain-flow effect at the advancing melt front (an outward trajectory of fluid particles from the central region to the bounding walls), the skin material at the melt front will show up at the region adjacent to the mold walls. This process continues until the cavity is nearly filled, with skin material appearing on the surface and the end of the part as
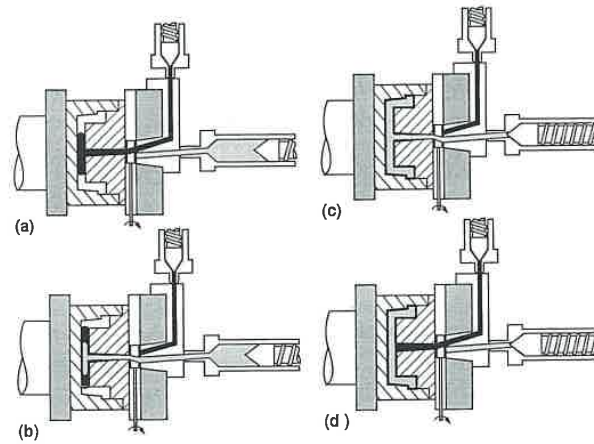

Figure 9.2 Sequential coinjection molding process [18] (Adapted from Ref. [21]. Reprinted by permission of the ASME International).

Table 9.2 Various Multicomponent Injection Molding Processes [29]

| Brief description	Machine/Mold requirements	Materials requirements	Typical applications
Coinjection Molding			
Two melts, injected sequentially or concurrently, flow into the cavity and separately form skin (material A) and core (material B) of the laminated part structure	Coinjection machine with special nozzle for mixture-free, concentric combining of the melt; alternative: three-way valve, staggered injection, material B penetrates within material A	Medium compatibility and melt viscosity requirement (skin material should have lower viscosity than the core material) Materials: PET, PA (reinforced and unreinforced), ABS (foamed and solid)	Headlight reflectors (core fiber reinforced, skin unreinforced to give good surface), Garden furniture (core foamed to reduce weight, skin solid to give good surface) Spectacle frame (core carbon fiber reinforced to generate high tensile strength, skin transparent to give good surface)
Two-Color Injection Molding			
A premolded component is placed in a second larger cavity and then encapsulated with the same, different colored material	With separately molded insert: standard molds and machines; More efficient set-up: tilting or transfer molds and two-color injection molding machine with two separate injection units	Low compatibility requirement; Materials: mostly ABS, PC, PA12, PA blends	Keyboard keys and push buttons for computers, telephones, and toys
Overmolding (Multi-Component Injection Molding, Injection Welding)			
A premolded component is placed in a second larger cavity and then encapsulated with a different, compatible thermoplastic	Same as above; It is advantageous to use "core-back" molds (with retractable slides) that permit overmolding of a hot insert using the same mold and a molding machine with two separate injection units	Maximum compatibility requirement; Materials: frequently PA12 or PA 12 copolymers, also PA/PU pairs	Hand-held power tools grips, damping elements in ski shoe shells, elastic inserts in safety and sport goggles, sight glasses in technical equipment
Bi-Injection Molding			
Two melts of different thermoplastics, guided separately (two runner systems, at least two gates) meet in the cavity	Machine with two separately controllable injection units; the melt front advancements determine the weld line location, high control quality needed	Medium compatibility requirement; greater number of pairs possible than overmolding	Flexible functional elements in an otherwise stiff part; positioning of core in thickenings by separate gates

(a)   (b)
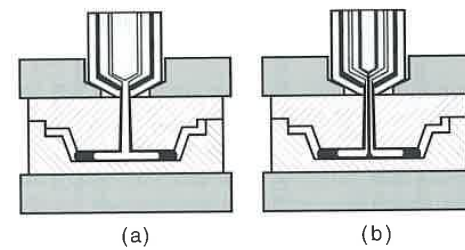
Figure 9.3 (a) Two-channel and (b) three-channel techniques [22].

shown in Fig. 9.2(c). Finally, a small additional amount of skin material is injected again to purge the core material away from the sprue so that it will not appear on the part surface in the next shot [cf. Fig. 9.2(d)].

When there is not enough skin material injected prior to the injection of core material, the skin material may sometimes eventually be depleted during the filling process and the core material will show up on portions of the surface and the end of the part that is last filled. Such "core surfacing" or "core breakthrough" is generally undesirable, although it may depend on the design requirement and final application.

There are other variations to the sequential (namely, skin–core–skin, or A–B–A) coinjection molding process described earlier. In particular, one can start to inject the core material while the skin material is being injected (i.e., A–AB–B–A). That is, a majority of skin material is injected into a cavity, followed by a combination of both skin and core materials flowing into the same cavity, and then followed by the balance of the core material to fill the cavity. Again, an additional small amount of skin injection will "cap" the end of the sequence, as described previously. In addition to the one-channel technique configuration, two- and three-channel techniques (cf. Fig. 9.3) have been developed that use nozzles with concentric flow channels to allow simultaneous injection of skin and core materials [22]. More recently, a new version of co-injection molding process that employs multi-gate coinjection hot-runner system has become available. Such a system moves the joining of skin and core materials into the mold, as shown in Fig. 9.4. In particular, this hot-runner system has separate flow channels for the skin and core materials. The two flow streams are joined at each hot runner coinjection nozzle. In addition to all the benefits associated with conventional hot-runner molding, this system allows an optimum ratio of skin and core in multi-cavity or single-cavity molds [23].

9.1.2   Process Advantages

Coinjection molding offers a number of cost and quality advantages, as well as design flexibilities and environmental friendliness as described later.
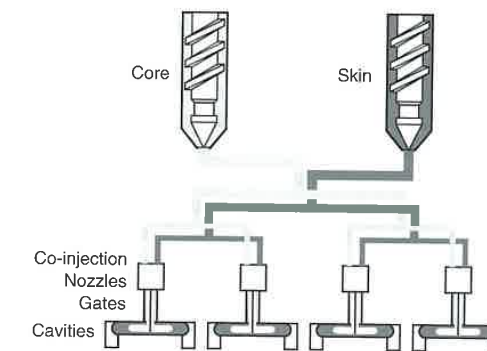
Figure 9.4 Multi-gate coinjection hot runner system with separate flow channels for skin and core materials, which join at each hot runner nozzle [23].

Material Cost Reduction and Recycling

High-performance and exotic engineering materials can be expensive but necessary for some applications. Coinjection provides the opportunity to reduce the cost of the product by utilizing lower-cost materials wherever the high-performance material is not necessary, perhaps in the core. That is, it permits the use of low-cost or recycled plastics as the core material, invisibly sandwiched within thin, decorative, expensive skin surfaces typically made of virgin plastic material. An example of this is a gear wheel (cf. Fig. 9.5) whose outer skin consists of a fluoropolymer or carbon fiber filled polyamide (nylon). As the recycling of postservice plastics becomes necessary by law in many countries, coinjection molding offers a cost-effective manufacturing technique to consume 100% recycled materials in high content.

Figure 9.5 Gear wheel (101.6 mm in diameter) with PTFE-filled white polyamide (nylon) 66 skin and glass-filled solid black polyamide 66 core made by coinjection molding [21].

Quality Surface with Foamed Core Material

In the case of thick-wall products, coinjection is preferred to conventional structural foam because of its superior cosmetic surfaces. Structural foam parts are often sanded, primed, base painted, and texture coated, all of which is expensive. A solid skin combined with a foamed core provides the advantages of structural foam, such as reduced part weight, low molded-in s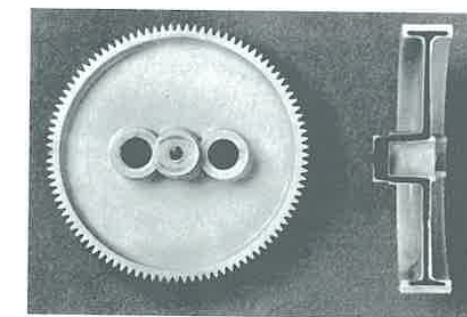tresses, straight sink-free parts, and design freedom, yet without the objectionable elephant-skin surface defects. Coinjection molding with foamed material also features an excellent weight-to-strength ratio and produces better performance than gas-assisted injection molding for sensitive/fragile polymer materials. For thin-wall parts, such as food packaging and bottles, coinjection also offers additional benefit in terms of physical and mechanical properties of the part and cost saving with foamed core.

Modification of the Part Quality and Property

With coinjection, one can obtain a combination of properties by joining different materials in one part, which is not available in a single resin. For instance, an elastomeric skin over a rigid core will provide a structure with soft touch. Another example includes a combination of a brittle material with a high-impact-resistant material, which provides excellent material properties. In applications where the performance of the components demands the use of reinforced materials, coinjection offers a solution that combines the aesthetic and property attributes of an unreinforced skin material with the benefits of a highly reinforced core material. Additional performance and cost improvements can be made by combining the conductive plastic with a more impact-resistant and less-expensive grade of plastic through co-injection molding. Such an application includes using either a skin or core polymer filled with a conductive material (e.g., aluminum flakes, carbon black, or nickel-coated graphite fibers) to provide the molded part (e.g., computer housings) with electromagnetic shielding (EMI) properties and grounding characteristics (cf. Fig. 9.6) [24].
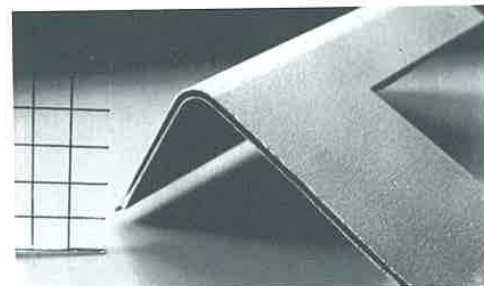

Figure 9.6 Section of a molding housing with an outer skin of ABS and a core of the same resin filled with 35 wt% of electrically conductive carbon black [24].

9.1.3   Process Disadvantages

Despite all the potential benefits of coinjection molding, the process has been slow to gain widespread acceptance for several reasons. First of all, the coinjection machine usually costs 50 to 100% higher than standard injection molding equipment [25]. This high investment cost offsets the benefits of developing unique processing techniques, improving part quality, and permitting the use of recycled materials. In addition, the development for a coinjection mold takes longer time than a conventional injection mold. This is also true for process set up, as the process requires additional control parameters for timing and controlling the injecting core material.

9.1.4   Applicable Materials

Coinjection molding can be employed for a wide variety of materials. Although most of the materials used are thermoplastic, there are some promising developments with using thermosetting materials, which are coinjected with thermoplastic materials. Because two materials are used in coinjection molding processing, the flow behavior (Fig. 9.2) and the compatibility of material properties are very important. In considering the material selection, the most important properties are viscosity difference and the adhesion between the skin and core material. Because the core material needs to penetrate the skin material in front of it, it is desirable to have a skin material with a viscosity lower than that of the core material. Using low-viscosity material in the core may cause the core flow front to travel too fast relative to the skin, which results in undesirable core surfacing. Experimental studies of coinjection molding have been conducted to examine the effect of relative viscosity ratio of skin and core materials on their spatial distribution within the part [26].

Because the materials are laminated together in the part, an effective adhesion of skin and core material is desirable for optimum performance. Table 9.3 provides some basic guidelines on a wide range of material combinations [27]. It should be noted that this information should only be used as a guide or benchmark. The actual performance must be determined by the application because the molding conditions and the operation/service conditions will influence the final performance. The other material property that needs to be of concern is material shrinkage. The rule of thumb is that materials with similar molding shrinkage should be paired in order to reduce stresses in the joining layers.

9.1.5   Typical Applications

Coinjection molding offers a technically and economically viable solution for a wide range of commercial applications in the emerging markets, which include automotive, business machines, packaging, electronic components, leisure, agriculture, and

Table 9.3   Material Adhesion Characteristics [25]

The table is a symmetrical adhesion matrix. Column and row headers (both axes): ABS, ASA, CA, EVA, PA 6, PA 66, PC, PE-HD, PE-LD, PMMA, POM, PP, PPO mod., PS-GP, PS-HI, PBTP, TPU, SAN, TPR, PETP, PVAC, PPSU, Blend PC-PBTP, Blend PC-ABS.

(From *Injection Molding Alternatives—A Guide for Designers and Product Engineers* (1998), Avery, J. A., Hanser Publishers, Munich, p.107, Table 5.4.1)

The symbols indicate: ■ = good adhesion; ▨ = poor adhesion; X = no adhesion.

From *Innovation in Polymer Processing: Molding*, Stevenson, J. F. (Ed.) (1996) Hanser, Munich, pp. 338–339, Table 9.2.

Courtesy of Battenfeld Gmbh, Meinerzhagen, Germany.

Standard plastic material abbreviations: ABS, acrylonitrile–butadiene–styrene; ASA, acrylic styrene acrylonitrile; CA, cellulose acetate; EVA, ethyl vinyl alcohol; PA 6 or nylon 6, polyamide 6; PA 66—nylon 66, polyamide 66; PC, polycarbonate; PE—HD, polyethylene—high-density; PE—LD, polyethylene—low density; PMMA, polymethylmethacrylate; POM, polyoxymethylene [(poly) acetal]; PP, polypropylene; PPO Mod., polyphenylene ether, modified; PS-GP, polystyrene—general purpose; PS-HI or HIPS, polystyrene: high-impact; PBTB or PBT, polybutylene terephthalate; TPU, thermoplastic polyurethane; PVC-W, polyvinyl chloride; SAN, styrene acrylonitrile; TPR, thermoplastic rubber; PETP or PET, polyethylene terephthalate; PVAC, polyvinyl alcohol; PVC, polyvinyl chloride; PPSU, polyphylene sulfide; Blend PC-PBTP, blend, polycarbonate–polybutylene–terephthalate; Blend PC-ABS, blend, polycarbonate–acrylonitrile–butadiene–styrene.

"soft touch" products. Example applications include canoe paddles, toilet seats and cisterns, computer housings, copier parts, cash register covers, television escutcheons, audio cabinets, circuitry and electronics enclosures, garden chairs, boxes and containers, shoes and soles, paint brush handles, metal hand-rims on wheelchairs, thin-wall containers and beverage bottles, automotive parts such as exterior mirror housings and interior door handles and knobs, components for high-end ovens, audio speaker housings, and music center mainframes.

Cost-reduction opportunities and the desire and mandated requirements to use recycled materials will drive more applications to coinjection in new and existing markets. More refined technology and a broader experience base among designers, processors and original equipment manufacturers (OEMs) will also expand the sporadic current use of the process. Useful design guidelines for coinjection molding and other multicomponent injection molding processes can be found in Ref. [28].

9.2   Fusible (Lost, Soluble) Core Injection Molding

The fusible (lost, soluble) core injection molding process produces complicated, hollow components with complex and smooth internal geometry in a single molding operation. This process is a form of insert molding in which plastic is injected around a temporary core of low melting-point material, such as tin-bismuth alloy, wax, or a thermoplastic. After molding, the core will be physically melted (or chemically dissolved), leaving its outer geometry as the internal shape of the plastic part. This process reduces the number of components required to make a final assembly or substitutes plastic for metal castings to boost performance (e.g., corrosion resistance) while saving weight, machining, and cost.

9.2.1   Process Description

Different techniques are available to produce single-piece components featuring complex, smooth internal geometry and a high dimensional stability, which cannot be obtained through the conventional injection molding process [1,30,31]:

- Fusible core technique
- Soluble core technique
- Salt core technique

All of these techniques employ the same principle: the production of injection molding with a lost core that gives the internal contour of the molded part. Among these lost core processes, fusible core technique is the most energy-intensive method. Nevertheless, this drawback is offset by the low core losses, smoother internal surface
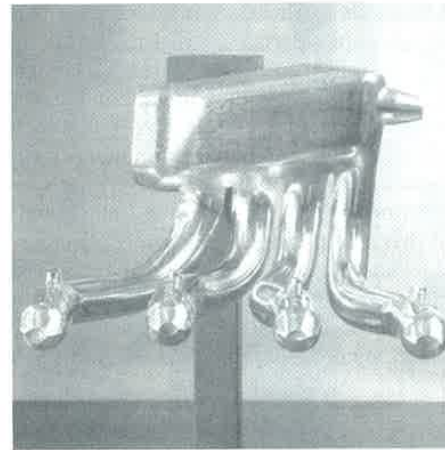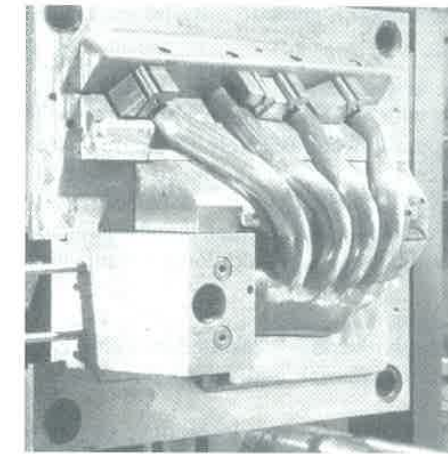
Figure 9.7 A eutectic bismuth-tin (BiSn 138) alloy core for an intake manifold [30]. Source: BASF. (From *Innovation in Polymer Processing—Molding*, Stevenson, J. F., (Ed.) (1996), Hanser, Munich, p. 159, Fig. 4.7.)

requiring low finishing cost, faster heat dissipation by using a stronger and highly conductive metal core. Fusible core injection molding basically comprises the following steps:

1. One or more core pieces are prefabricated at a separate station from the injection molding machine (cf. Fig. 9.7).
2. If more than one core piece is needed, the core assembly will take place in order to make a single-unit core.
3. The core then goes through a preheating process to reach temperature uniformity throughout the core before it is inserted into the mold (cf. Fig. 9.8).
4. Plastic is injected over the core in the injection machine mold, flowing evenly over the core without solidifying or melting the core.
5. After molding, the plastic part is taken with the core to a "melting" station, where the core is melted (or dissolved) out of the hollow plastic component (cf. Fig. 9.9).
6. The plastic part is cleaned with water and/or a cleaning solution and then taken to the next step for postmolding operations while the core material is recycled.
7. These postmolding operations include additional drilling or surfacing. In the end, a single-unit, hollow plastic part is obtained.

To minimize the core loss due to oxidation and energy consumption, a number of eutectic alloys have been found to be suitable for fusible core injection molding. These eutectic alloys exhibit stable dimensional stability (with low expansion or shrinkage), a sharp melting point, and a rapid phase transition. It is apparent that the

Figure 9.8 A eutectic bismuth-tin (BiSn 138) alloy core secured in the mold for an intake manifold [30]. Source: BASF. (From *Innovation in Polymer Processing—Molding*, Stevenson, J. F., (Ed.) (1996), Hanser, Munich, p. 164, Fig. 4.12.)

melting temperature of the fusible alloy should be lower than that of the overmolding plastic so that it can be melted out in the melting bath. Nevertheless, due to the high thermal diffusivity as well as its massive thermal inertia and latent heat of the alloy, the surface temperature of the core remains below its melting point during molding. This phenomenon is akin to the relatively small temperature rise at the mold-wall in conventional injection molding where the high thermal conductivity of the metal mold rapidly removes the heat from the molten plastic.

Figure 9.9 (Left) An intake manifold made from glass-fiber-reinforced polyamide 66 by fusible core technique; (center) core overmolded with plastic; (right) fusible core cast from bismuth-tin alloy [30]. Source: BASF. (From *Innovation in Polymer Processing—Molding*, Stevenson, J. F., (Ed.) (1996), Hanser, Munich, p. 155, Fig. 4.3.)

9.2.2   Process Advantages

The primary advantage of fusible core injection molding is the ability to produce single-piece plastic parts with highly complex, smooth internal shapes without a large number of secondary operations for assembly. Compared with options of aluminum casting and machining, cost savings with fusible core injection molding are claimed to be up to 45% and weight savings can be as high as 75% [25]. Plastic injection molding tools also have a much longer life than metal casting tools due to the absence of chemical corrosion and heavy wear. Additional advantages include:

- greater freedom in design
- high surface quality with no zones weakened by welding or joining
- dimensionally accurate internal and external contours and high structural integrity
- low labor intensity with only moderate levels of secondary operations (trimming, drilling, finishing, etc.)
- easier mold design
- little scrap in the overall process
- integration of inserts.

9.2.3   Process Disadvantages

The main disadvantage of fusible core injection molding is cost. First, it requires a significant capital investment, ranging from $6 to 8 million [25]. The fusible core molding unit also requires a very large injection molding machine, casting machine and equipment, a "melt-out" station, and robotic handling devices for the typically heavy plastic molding with core. There are also significant operating expenses, which come from core casting and electricity usage resulting from more machines being used as well as from core casting and melting. Another disadvantage is mold and machine development for casting and injection molding, which take time due to complex design and prototyping requirements. Because of this cost factor, the fusible core injection molding process for nylon air intake manifolds has been tempered by a shift to less-capital-intensive methods such as standard injection molding, where part halves are welded (twin-shell welding) or mechanically fastened [32].

9.2.4   Applicable Materials

Fusible core injection molding is generally employed for engineering thermoplastics, primarily glass fiber reinforced polyamide (PA) 6 and polyamide 66. Other materials used are glass fiber reinforced polyphenylene sulfide (PPS) and polyaryletherketone (PAEK); glass reinforced polypropylene (PP); and glass reinforced

Table 9.4   Various Fusible Core Injection Molding Applications [25]

Automotive applications	Nonautomotive applications
Air-intake manifolds	Bearing housings
Air-intake manifold seat flanges	Thermostat valves
Fuel rails	Water valves
Power steering inlet pipe adapters	Water pump impellers and housings
Turbo charger housings	Coffee machine and showerhead feed pipes
Thermostat housings	Plumbing fixtures
	Tennis racquets
	Canoe paddles
	Bicycle wheels
	Wheelchair wheels
	Helicopter components

polyoxymethylene (POM) [25]. Low-profile thermosets have reportedly been used in fusible core injection molding, which require a core with a higher melting point [30]. A number of factors must be considered in selecting the plastic materials. For example, the plastic component must be compatible with the core alloy and be able to withstand melting bath temperatures, which ranges from 100 to 180°C, in addition to typical considerations of the operation environment and mechanical property requirements.

9.2.5   Typical Applications

The fusible core injection molding has so far been used primarily for automotive air intake manifolds (see Fig. 9.9). Improved processing engineering and greater design freedom, however, will open up innovative and interesting applications for years to come. Other typical applications of fusible core injection molding are listed in Table 9.4.

9.3   Gas-Assisted Injection Molding

The gas-assisted injection molding process consists of a partial or nearly full injection of polymer melt into the mold cavity, followed by injection of inner gas (typically nitrogen) into the core of the polymer melt through the nozzle, sprue, runner, or directly into the cavity. The compressed gas takes the path of the least resistance flowing toward the melt front where the pressure is lowest. As a result, the gas penetrates and hollows out a network of predesigned, thick-sectioned gas channels, displacing molten polymer at the hot core to fill and pack out the entire cavity. The

essentially inviscid gas ideally transmits the gas pressure effectively as it penetrates to the extremities of the part, thereby requiring only a relatively low gas pressure to produce sufficient packing and a fairly uniform pressure distribution throughout the cavity. As a result, this process is capable of producing lightweight, rigid parts that are free of sink marks and have less tendency to warp.

9.3.1   Process Description

Gas-assisted injection molding starts with a partial or nearly full injection of polymer melt, as in conventional injection molding, followed by an injection of compressed gas. The compressed gas is usually nitrogen because of its availability, low cost, and inertness with the polymer melt. During the gas-injection stage, gas penetrates the hot core of the thickest sections where the material remains molten and fluid, displacing it to fill the extremities of the cavity and pack out the mold.

In the so-called gas-pressure control process, the compressed gas is injected with a regulated gas pressure profile (constant, ramp, or step). In the "gas-volume control" process, gas is initially metered into a compression cylinder at preset volume and pressure; then, it is injected under pressure generated from reducing the gas volume by movement of the plunger. Conventional injection molding machine with precise shot volume control can be adapted for gas-assisted injection molding with add-on conversion equipment, a gas source, and a control device for gas injection, as shown in Fig. 9.10. Gas-assisted injection molding, however, requires a different approach to product, tool, and process design due to the need for control of additional gas injection and the layout and sizing of gas channels to guide the gas penetration in a desirable fashion. As an illustration, Fig. 9.11 shows the schematic of gas injecting through
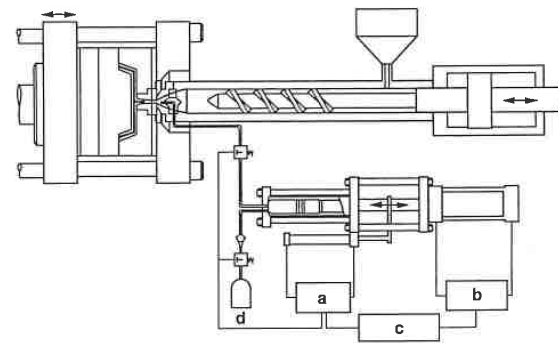
Figure 9.10 Schematic of a typical injection molding machine adapted for gas-assisted molding with add-on gas-compression cylinder and accessory equipment. (a) Electrical system, (b) hydraulic system, (c) control panel, (d) gas cylinder [34]. (From *Innovation in Polymer Processing—Molding*, Stevenson, J. F., (Ed.) (1996), Hanser, Munich, p. 46, Fig. 2.1.)
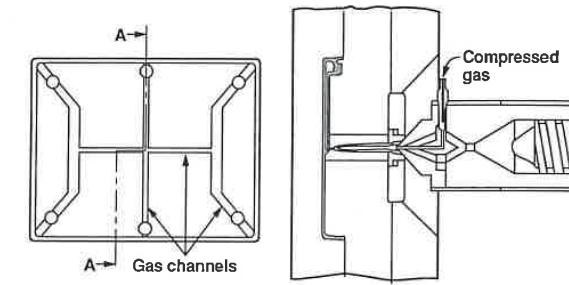
Figure 9.11 Schematic showing injection of gas through the nozzle and the gas distribution in the rib of the molding with a network of thick-sectioned gas channels [34]. (From *Innovation in Polymer Processing—Molding*, Stevenson, J. F., (Ed.) (1996), Hanser, Munich, p. 46, Fig. 2.2.)

the nozzle and the gas distribution in the rib of the molding with a network of thick-sectioned gas channels.

Process Cycle

As illustrated in Fig. 9.12, the gas-assisted injection molding process begins with a resin-injection stage (between Points 1 and 2), which is the same as the conventional
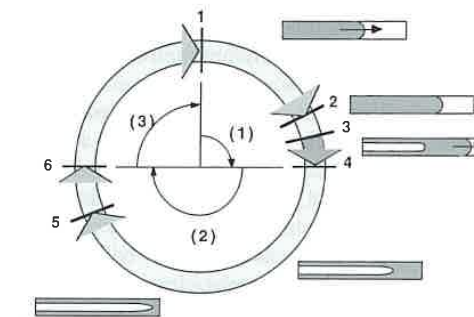
Process Cycle:

Stage (1) Filling Stage
1:   Cycle begins
1-2: Resin injection stage
2:   Resin injection ends
2-3: Delay time
3:   Gas injection begins
3-4: Gas injection during filling
4:   Cavity filling completed

Stage (2) Post-Filling Stage
4-5: Gas pressure held
5:   Gas pressure released
6:   Mold opens

Stage (3) Mold Opening Stage

Figure 9.12 Process cycle of gas-assisted injection molding. Note that resin-injection time is between Points 1 and 2, the timer starts gas injection at Point 3, the fill time is between Points 1 and 4, and gas-injection time is from Points 3 to Point 5 [34]. (From *Innovation in Polymer Processing—Molding*, Stevenson, J. F., (Ed.) (1996), Hanser Publishers, Munich, p. 48, Fig. 2.3.)

injection molding process. Before the gas injection is triggered at Point 3, there is an optional delay time during which the polymer at the thin section cools so that the incoming gas can only core out the designated thick sections that serve as the gas channels. The gas injection can sometimes take place before the end of the resin injection to avoid hesitation marks at the melt front location during the gas delay time. It can be done, however, only when the gas-injection point is different from the polymer entrance; otherwise, simultaneous injection of polymer and gas at the same entrance will result in an undesirable, corrugated polymer skin thickness. The gas-injection time is the duration over which gas pressure is imposed at the gas entrance(s) (front Point 3 through Point 5). Because gas is injected to help fill and pack out the mold, the gas-injection stage actually consists of both gas-injection-filling and postfilling phases. The timer for gas injection and the gas-injection time are two important processing parameters that strongly influence the gas penetration. As in conventional injection molding, the part should be allowed to cool sufficiently inside the mold before the gas pressure is released at Point 5. The polymer skin thickness (also called residual wall thickness) over the cored-out gas channels ideally permits a significant reduction of cooling time. After that, the mold opens at Point 6 for the part to be ejected.

Although the total gas-injection time takes a significant portion of the entire cycle, the actual elapsed time from the introduction of the gas to the instant when the whole cavity is filled is very short, as illustrated in the process cycle clock in Fig. 9.12. Nevertheless, this relatively short gas-injection-filling stage is crucial to the success of the molded part because the various molding problems associated with this process, such as air trap, gas blow-through, short shot, gas permeation into thin section, uneven gas penetration, and the like, typically take place in this stage. The gas penetration created by displacing the polymer melt during the gas-injection-filling stage is often referred to as the *primary gas penetration*. This is typically the stage when the majority of gas penetration is determined (assuming a short shot of resin is injected), which impacts the quality of the final products. After the cavity is filled, a certain level of gas pressure is maintained to pack out the molded part. During this postfilling stage (or the holding stage), the so-called secondary gas penetration continues to compensate for the material shrinkage, primarily along the gas channels or thick sections. When the molded part cools down and becomes rigid enough inside the mold, the gas pressure is released prior to the ejection of the part. The essentially inviscid gas ideally transmits the gas pressure effectively as it penetrates to the extremities of the part, resulting in a uniform pressure distribution throughout. This process is consequently capable of producing lightweight, rigid parts that are free of sink marks and have less tendency to warp. This process provides tremendous flexibility in the design of plastic parts. A comprehensive review of the technology and applications can be found in Refs. [33,34] and the cited references.

Special Gas-Assisted Injection Molding Processes

Because of the versatile and promising capabilities of this process, some alternative gas-assisted injection molding processes have been developed and become commer-

cially available. For example, instead of using the compressed nitrogen, the "liquid gas-assist process" injects a proprietary liquid into the melt stream. This liquid is converted to a gas in a compressed state with the heat of the polymer melt [35]. After the filling and packing of the cavity, the gas is absorbed as the part cools down, thereby eliminating the need to vent the gas pressure. In the so-called water-assisted injection molding process, water, which does not evaporate during displacement of the melt, is injected after resin injection. Compared with "conventional" gas-assisted injection molding, water facilitates superior cooling effect and thus a shorter cooling time, as well as thinner residual wall thickness and larger component diameters [36]. The "partial frame process" injects compressed gas into the strategically selected thick sections to form small voids of 1 to 2 mm (0.04 to 0.08 in) in diameter. Such a process can be employed to reduce sink marks and residual stresses [37] over the thick sections. Finally, the "external gas molding process" is based on injecting gas in localized, sealed locations (typically on the ejector side) between the plastic material and the mold wall [38,39]. The gas pressure maintains contact of the cooling plastic part with the opposite mold wall while providing a uniform gas pressure on the part surface supplementing or substituting for conventional holding pressure. This kind of process is suitable for parts with one visible surface where demands on surface finish are high and conventional use of gas channels is not feasible.

9.3.2   Process Advantages

The characteristics of gas-assisted injection molding lead to both the various advantages and the inherent difficulties associated with the design and processing. Recall that gas normally takes the path of least resistance as it flows toward the polymer melt front while hollowing out the thickest sections. To facilitate the gas penetration in a desirable pattern, the molded part typically has a network of built-in gas channels that are thicker than the main wall section. The gas ideally penetrates and hollows out only the network of gas channels by displacing the molten polymer melt at the hot core to fill the entire cavity.

Figure 9.13 schematically illustrates the mechanism of how the polymer melt and gas interact in the cavity. It is well known that in the conventional injection molding process, the pressure required to advance the polymer increases with the amount of polymer injected (or, equivalently, the flow distance). Figure 9.13(a) schematically depicts the rise in pressure during filling of the mold cavity without gas injection. It should be noted that the gap-wise averaged melt velocity is proportional to the magnitude of the pressure gradient and the melt fluidity. As the flow length of the polymer melt increases, therefore, the inlet pressure has to increase to maintain a certain pressure gradient if the flow rate is to be kept constant.

On the other hand, the pressure requirement with gas-assisted injection molding is the same as the conventional process during the resin-injection stage. Upon introduction of gas into the cavity, the gas starts to displace the viscous polymer melt, pushing it to fill the extremities of the cavity. Because the gas is essentially inviscid,
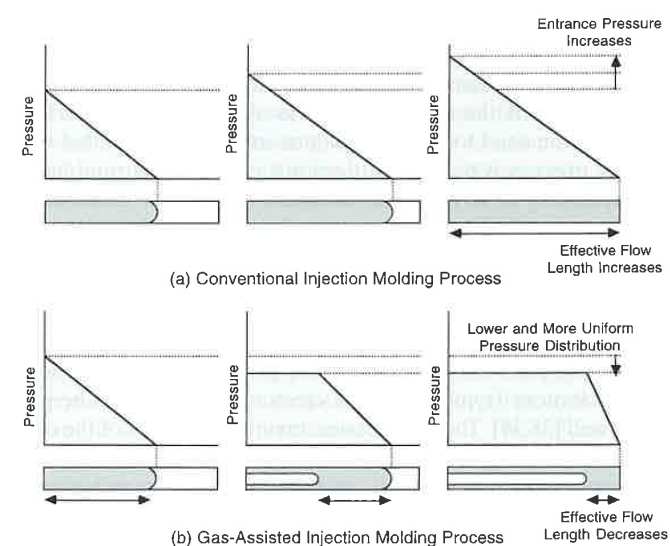
Figure 9.13 (a) The evolution of pressure distribution for the conventional injection molding process. (b) The evolution of pressure distribution for the gas-assisted injection molding process. Note the lower but more uniform pressure distribution in case of gas-assisted injection molding [34]. (From *Innovation in Polymer Processing—Molding*, Stevenson, J. F., (Ed.) (1996), Hanser Publishers, Munich, p. 49, Fig. 2.4.)

it can effectively transmit the gas pressure, without a significant pressure drop, to the advancing gas–melt interface [cf. Fig. 9.13(b)]. As the gas advances toward the melt front, therefore, the pressure required to keep the melt ahead of the gas moving at the same velocity decreases, because the effective flow length decreases. The gas pressure required to fill the mold cavity can consequently be lower than the required entrance melt pressure for the conventional injection molding process. Further, the resulting pressure distribution is more uniform in a gas-injected part, which induces less residual stresses as the polymer cools down during the post-filling stage.

The gas-assisted injection molded part can therefore be produced with a lower gas pressure requirement (which normally leads to lower clamping tonnage) and has less tendency to warp. Because the part is internally pressurized with gas during the packing stage, all shrinkage is taken up on the inside of the part, which eliminated the sink marks on the surface of the part. Other advantages of gas-assisted injection molding include:

- Increased part rigidity due to enlarged cross-sections at hollowed gas-channels, especially for large structural parts (also called open channel flow parts)
- Reduced material consumption, especially for rodlike parts (also called contained channel gas flow parts)

- Reduced cooling time since the thick sections are cored out
- Greater design flexibility to incorporate both thick and thin sections in the same part for part consolidation

9.3.3   Process Disadvantages

Because gas-assisted injection molding involves dynamic interaction of two dramatically dissimilar materials flowing within typically complex cavities, the product, tool, and process designs are quite complicated. Previous experience with the conventional injection molding process is no longer sufficient to deal with this process, especially with injecting gas and controlling its penetration pattern. There will therefore, be a learning curve before one can fully benefit from this process. The design process, more specifically, has to be carried out with the process physics in mind in order to avoid problems associated with the dynamic nature of this process. In addition, the start up investment of gas-assisted injection molding includes both the cost of additional gas injection unit and source of nitrogen as well as the cost of licensing if a patented process is being used.

Typical molding problems associated with gas-assisted injection molding include:

- air trapping
- gas permeation into thin sections
- uneven gas penetration
- gas blow-through
- surface defects
- short shot
- surface blisters

9.3.4   Applicable Materials

Most of the thermoplastic materials, technically speaking, can be used for gas-assisted injection molding. Gas-assisted injection molding has been extended to thermosetting polyurethane [40] and powder injection molding [41]. Note that the material selection should be based on requirements of application performance, such as stiffness, chemical resistance, and material strength at the operating temperature. It has also been reported that the reproducibility of the polymer skin thickness is normally poor for polymers with a strong shear-thinning viscosity [42].

9.3.5   Typical Applications

Typical applications for the gas-assisted injection molding process can be classified into three categories, or some combination of them:

(a)
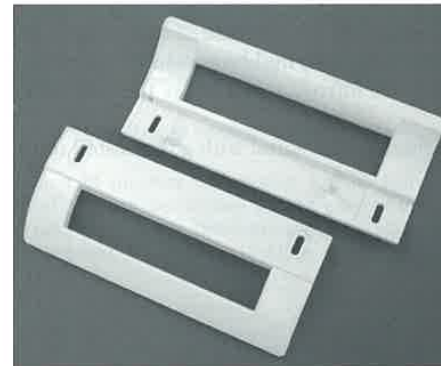
(b)

Figure 9.14 Gas-assisted door handles. [Parts courtesy of Battenfeld of America (a), West Warwick, Rhode Island, USA and Cinpres Limited (b), Tamworth, England.]

- Tube- and rodlike parts, where the process is used primarily for saving material, reducing the cycle time by coring out the part, and incorporating the hollowed section with product function. Examples are clothes hangers, grab handles (cf. Fig. 9.14), chair armrests, shower heads, and water faucet spouts.
- Large, sheetlike, structural parts with a built-in gas-channel network, where the process is used primarily for reducing part warpage and clamp tonnage as well as to enhance rigidity and surface quality. Examples are automotive panels, business machine housings, outdoor furniture (cf. Fig. 9.15), and satellite dishes.
- Complex parts consisting of both thin and thick sections, where the process is used primarily for decreasing manufacturing cost by consolidating several assembled parts into one single design. Examples are automotive door modules (Fig. 9.16), television cabinets, computer printer housing bezels, and automotive parts.
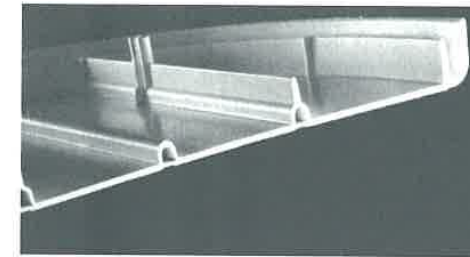
Figure 9.15 Cross-section of a garden tabletop. (Courtesy of Steiner Company, Austria.)

General design guidelines aimed at providing basic know-how in design and manufacturing of parts using gas-assisted injection molding can be found in the literature [34]. In addition, computer simulation for gas-assisted injection molding has also become available to help engineers gain process insights and make rational design decisions.

Figure 9.16 Automotive door modules (Courtesy of Delphi Interior & Lighting Systems) [25]. (From *Injection Molding Alternatives—A Guide for Designers and Product Engineers*, Avery, J. A., (1998), Hanser, Munich, p. 97, Fig. 5.3.18 and p. 98, Fig. 5.3.19.)

9.4   Injection-Compression Molding

Injection-compression molding (ICM) is an extension of conventional injection molding by incorporating a mold compression action to compact the polymer material for producing parts with dimensional stability and surface accuracy. In this process, the mold cavity has an enlarged cross-section initially, which allows polymer melt to proceed readily to the extremities of the cavity under relatively low pressure. At some time during or after filling, the mold cavity thickness is reduced by a mold closing movement, which forces the melt to fill and pack out the entire cavity. This mold compression action results in a more uniform pressure distribution across the cavity, leading to more homogenous physical properties and less shrinkage, warpage, and molded-in stresses than are possible with conventional injection molding.

9.4.1   Process Description

ICM is basically the same as conventional injection molding in the initial filling stage. After a preset amount of polymer melt is fed into an open cavity, a mold compression action is engaged and continues to the end of the molding process. Figure 9.17 illustrates both the initial injection stage and the subsequent compression stage. The compression can also take place when the polymer is being injected. Based on the process variation, ICM can be further classified into the following three categories [25]:

- Two-stage sequential ICM
- Simultaneous ICM
- Selective ICM
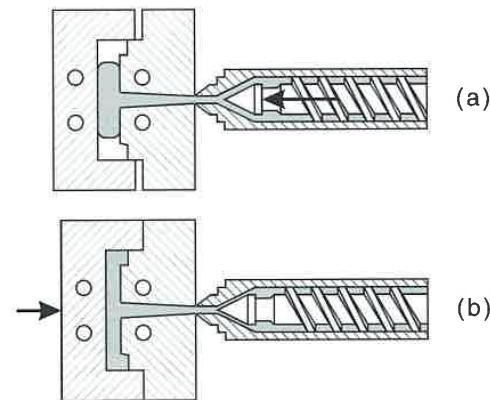
Figure 9.17 Typical injection-compression molding process sequence.

As the simplest type of these three techniques, the two-stage sequential ICM consists of separate injection stage and compression stage. During the injection stage, resin is injected into a cavity whose thickness is initially oversized by 0.5 to 10 mm greater than the nominal thickness. At the end of the resin injection, compression stage begins during which the mold cavity thickness is reduced to its final value. The mold compression action forces the resin to fill the rest of the cavity and, after the cavity is completely filled, provides packing to compensate for the material shrinkage due to cooling. A potential drawback associated with the two-stage sequential ICM is the "hesitation" or "witness" mark resulting from flow stagnation during injection-compression transition.

To avoid this surface defect and to facilitate continuous flow of the polymer melt, the simultaneous ICM activates mold compression while resin is being injected. The third technique, the selective ICM method, starts with the cross-section at the final nominal value. During the injection stage, melt pressure drives the mold back toward the cylinder, which is mounted to an unpressurized movable core. Based on cavity pressure or time, the compression stage is activated by pressurizing the cylinder to force the part-forming surface on the movable core to compress the melt [25].

A typical injection-molding machine with precise shot-volume control can be adapted for ICM; however, an additional control module is required for the mold compression stage. In addition, there are some requirements and suggestions for the injection-compression process and the molds [43]:

- A vertical flash face design is necessary to prevent uncontrolled leakage of the melt into the mold partition surface.
- A needle valve nozzle must be used to seal the cavity mechanically and to keep the compression delay time (from end of injection to beginning of compression) as low as possible to avoid hesitation marks.
- A mechanical stop is required in the hot runner system to ensure a precise metering of the melt to be injected into the mold. This way, it is possible to achieve a constant shot weight.

9.4.2   Process Advantages

The primary advantage of ICM is the ability to produce dimensionally stable, relatively stress-free parts, at a low pressure, clamp tonnage (typically 20 to 50% lower), and reduced cycle time. Recall that in conventional injection molding, a high injection and packing pressure has to be applied at the machine nozzle to produce sufficient pressure level at the extremities of the cavity to drive the flow and pack out the material. For thin-walled applications, such as compact disks, there is typically a significant pressure variation across the part due to high flow resistance. Such a high-pressure variation results in nonuniform packing and volumetric shrinkage within the part, leading to molded-in residual stresses and severe part warpage. With ICM the packing pressure is now applied in the thickness direction (as opposed to crossing

Table 9.5   Comparisons of Different Processes for Production of Thin-Walled Parts

Technology	Advantages	Drawbacks
Stretch-forming and vacuum thermoforming	Simple molds Low investment	Waste due to edge trim Wall thickness determined by the process
Dipping in solutions	Simple molds Low investment	Suitable for few plastics. Solvent vapors may be released Wall thickness determined by the process
Injection-compression molding	Any wall thickness possible Suitable for many plastics Short cycle times Good automation potential	High machine and mold costs Economic for high-volume products

the part dimension) for majority (if not all) of the part. As a result, ICM permits a lower and yet much more uniform pack/hold pressure distribution that can effectively pack out the mold and minimize molded-in residual stresses and part warpage.

Table 9.5 compares the ICM with other two processes commonly used for the production of thin-walled parts.

9.4.3   Process Disadvantages

The process disadvantages of ICM can be summarized as follows:

- The molds for this process are relatively expensive and subject to high wear during the compression stage.
- Additional investment is required for the injection molding machine; namely, the control module for the compression stage.
- This technology is only profitable for high-volume products, such as compact disks, or products that require minimum molded-in residual stresses, such as optical lenses.

9.4.4   Applicable Materials

For thin-wall applications, difficult-to-flow materials, such as polycarbonate and poly-etherimide, have been molded to 0.5 mm [25]. On the other hand, high melt-flow-index polycarbonates are the most suitable materials for compact disks. In addition, most of the lenses are produced with polycarbonate due to its excellent optical properties. Other materials used in ICM include: acrylic, polyethylene, PPE/PA blends, and polypropylene, as well as thermoplastic rubber and most thermosetting materials [25].

9.4.5   Typical Applications

The ICM is the most suitable technology for the production of high-quality and cost-effective CD-Audio/ROMs. The ICM is also an appropriate technology for the production of many types of optical lenses. Figure 9.18 shows some of the typical ICM applications [44]. There has been a renewed interest in ICM for molding of thin-walled parts and for in-mold lamination (see Sec. 9.5).

9.4.6   Computer Simulation for Injection-Compression Molding

Several computer programs have been developed to simulate the ICM process (see, e.g., Refs. [45–47]). They use a Hele-Shaw fluid flow or 2.5-diameter-model to predict the melt front advancement and the distribution of pressure, temperature, and flow velocity dynamically during the injection melt filling, compression melt filling, and postfilling stages of the entire process. Based on the simulated results, it was found that ICM both shows a significant effect on reducing part shrinkage and provides much more uniform shrinkage within the molded part. The results have also shown that birefringence becomes smaller as the melt temperature increases, and as the closing velocity of the mold decreases. The flow rate and the mold temperature do not affect birefringence significantly. As far as the density distribution is concerned, the mold-wall temperature affects this parameter, especially near the wall. The flow rate, melt temperature, and mold closing velocity have insignificant effects on the density distribution [46].

Figure 9.18 Typical injection-compression molding applications [44].

9.5   In-Mold Decoration and In-Mold Lamination

In-mold decoration (IMD) comprises insertion of a film or foil into the cavity, followed by injection of polymer melts on the inner side of the insert to produce a part with the final finish defined by the decorative film/foil. On the other hand, the in-mold lamination process resembles IMD except that a multilayered textile laminate is used. These two processes provide a cost-effective way to enhance and/or modify the appearance of product for marking, coding, product differentiation, and model change without costly retooling. The laminated component can have desirable attributes (e.g., fabric or plastic skin finish with soft-touch) or properties (e.g., electromagnetic interference, EMI, or radiofrequency interference, RFI, shielding).

9.5.1   Process Description

IMD

In the IMD process, a predecorated carrier laminated onto film stock from the roll is pulled through the mold and positioned precisely between the mold halves. The film stock may be decorated by printing methods (e.g., silk-screen or hot stamping) prior to molding [48]. During the molding stage, the polymer melt contacts the film and fuses with it so that the decoration can be lifted off from the carrier film and strongly attach to the surface of the molding. An injection molding machine with a typical IMD set up is shown in Fig. 9.19.

In one of the IMD techniques, the so-called paintless film molding (PFM) or laminate painting process, a three-layer coextrusion film with pigment incorporated into
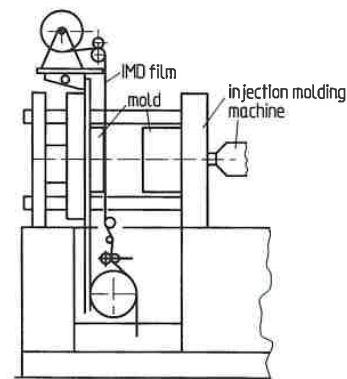
Figure 9.19 In-mold decorating setup with a foil-feeding device built into the injection molding machine [49]. (From *Injection Molding—An Introduction*, Pötsch, G. and Michaeli, W. (1995), Hanser, Munich, p. 175, Fig. 8.6.)

layers of clear-coat cap layers and core layer is first thermoformed into the shape of the finished part and then inserted into the cavity and overmolded with thermoplastics to produce a final part [50]. With this technique, it is possible to obtain a high-quality, extremely smooth paint finish on thermoplastic exterior body claddings and moldings ready for assembly without subsequent spray painting or finishing. Unlike in-mold coating and mold-in-color, this process provides high-gloss metallic and non-metallic finishes. It provides unique patterns and designs that are not feasible with paint. The paint laminate finish provides superior weatherability, acid etch resistance, and a safe worker environment because it is virtually pollution-free.

In addition to injection molding, IMD can be used with a variety of other processes, such as structural foam injection molding, ICM, compression molding, blow molding, thermoforming, resin transfer molding, and rotational molding [25].

In-Mold Lamination Process

Instead of using a thin film/foil as does IMD, the in-mold lamination process employs a multilayered textile laminate positioned in the parting plane to be overmolded by the polymer melt on the inner side. The decorative laminate can be placed in the mold as a cut sheet, pulled from the roll with needle gripper, or, by means of a clamping frame method. By means of a thermoforming operation, the clamping frame method allows defined predeformation of the decorative laminate during the mold-closing operation. In-mold lamination is also known as "laminate insert molding" [51] or "fabric molding" [52] for manufacturing automotive instrument panels and interior panels, respectively.

For in-mold lamination, the outer, visible layer of the decorative laminate can be made of polyester, PA, PP, polyvinyl chloride (PVC), or acrylonitrile-butadiene-styrene (ABS) film, cotton textile (woven, knitted, tufted, or looped fabrics), or leather. This outer layer typically comes with a variety of features to create an appearance or feel. In general, low surface texture is preferred because the ironing effect typically occurs with high pile or fiber loops. To provide the product with a soft-touch effect, there is typically an intermediate layer of polyurethane (PU), PP, PVC, or polyethersulfone (PES) foam between the top layer and the liner layer. Underneath the foam layer is the liner layer, which is used to stabilize the visible layer against shear and displacement, prevent the penetration of polymer melt into the intermediate layer, and provide thermal insulation against the polymer melt. This liner layer can be woven, knitted, or a nonwoven fabric. The typical structure of the in-mold laminated parts is shown in Table 9.6.

To avoid damage or undesirable folding of the laminate during molding, low injection pressure and low temperature are desirable. This makes low-pressure injection molding (see Sec. 9.8), ICM (see Sec. 9.4), compression molding, and cascade injection molding with sequential valve-gate opening and closing suitable candidates for in-mold lamination. Note that the latter process also eliminates the common problem associated with the weld lines, which are more evident with the IMD process.

Table 9.6 Typical Structure of the In-Mold Laminated Parts [53,54]

Structure	Typical dimension	Materials	Function
Coating (outer) material	Foil: 0.25 to 1 mm Textile: 0.25 to 4 mm	Foil: PVC, ABS, TPO Textile: PET, PA, PP	Decoration Tactile properties
Foam layer	1 to 3 mm	PU, PVC, PP	Soft touch Thermal insulation

Liner	0.5 to 1 mm	PET, PP	Thermal insulation,
			  stabilizing coating
			  material, increased tear
			  resistance
			  improved adhesion

Base (carrier, substrate) plastic material	2 to 3 mm	PP, PP-talc reinforced, ABS, ABS/PC blend	Define shape, give rigidity to part

9.5.2   Process Advantages

When compared with conventional surface decoration methods, such as painting, metallizing, hot stamping, PVC film laminating, and various painting methods, IMD and in-mold lamination offer a wide range of advantages, such as [55]:

- The process is a cost-effective way of surface decoration that replaces the traditional multistaged lamination process with a single molding cycle. The potential savings can amount up to 15 to 25%.
- The process is environmentally friendly with no volatile solvents released during adhesive lamination, and no posttreatment is needed.
- The decoration exhibits strong adhesion with the molding, as well as high surface-wear resistance and good chemical resistance.
- Use of the regrind is possible if permitted by the application and requirements.
- Three-dimensional decoration is possible within the limitations of the deformability of the film/foil and laminate.

9.5.3   Process Disadvantages

The disadvantages of IMD and in-mold lamination processes are additional equipment cost as well as extra steps for handling, die-cutting, performing, and placement of the decorative film/foil and laminate into the mold. Although conventional injection molding machines can be used with nonsensitive decorative materials, special machine equipment specifically made for these processes produces the best results and savings. Other disadvantages include longer cycle time (due to the insulating

effect of the decorative layer), part warpage that results from unbalanced cooling, part rejection associated with damage, creasing, folding, shade changes, overstretching, and weld-line marks of the decorative layer.

9.5.4   Mold Design and Processing Considerations

To ensure the quality of the molded parts with decorative laminate, special considerations on designs of part and mold, selection of coating and base (carrier) materials, and set up of process conditions have to be taken into account, as we will discuss in the following sections.

Mold Design Considerations

Although the existing tools can often be adopted for IMD and in-mold lamination, it is generally recommended to design the tool specifically for these processes. In addition to the conventional design rules, such as appropriate radii and taper angle, the following design considerations have to be taken into account:

- For IMD, the film/foil must be carried into the mold between well-separated dowel pins, and run close against (but without rubbing) the mold surface.
- The design of the mold and the way the decorative laminate is fixed in the mold should ensure that the material is not overstretched during mold closing.
- Complex surfaces may result in problems caused by air entrapment or stretching of the film and laminate.
- The weld lines and sink marks associated with ribs become more evident, especially with thin films and textiles due to accumulation of decoration material at weld lines.
- The injection of polymer melt must be carried out in such a way that it will not adversely displace or deform the decorative laminate, resulting in creasing of the decoration, or damaging the surface structure of the film/foil or the three-dimensional structure of the textile.
- Because the decorative laminate typically sits on the moving platen, the part must be ejected from the sprue side to avoid leaving ejector pin mark on the decorated side, which faces the moving half of the mold.
- For IMD, the film/foil must be wide enough to cover all the cavities if a multi-cavity system is used.
- When decoration is three-dimensional, the extensibility of the IMD carrier film must not be exceeded.
- Venting should be provided between the film/foil and the moving mold half to prevent entrapped air and thus burning during the molding.
- Care must be taken to ensure that no melt reaches the display side of the decorative laminate, either through it or around the edges of the blank.
- The construction of the mold should allow automatic insertion of the decorative laminate and removal of the overmolded part.

Processing Considerations

The process conditions have to be changed slightly when running with IMD or in-mold lamination. For example, to avoid undesirable displacement of the decorative laminate, especially as melt first contacts the insert, the initial injection speed should be low. In addition, the injection rate should be set in such a way that the required injection pressure is at minimum. Note that for injection molding, if the injection pressure required to fill a cavity is plotted against the fill time, a U-shape curve typically results, with the minimum value of the required injection pressure occurring at an intermediate fill time. The curve is U-shaped because, on the one hand, a short fill time involves a high melt velocity and thus requires a higher injection pressure to fill the cavity. On the other hand, the injected polymer cools more with a prolonged fill time. This leads to a higher melt viscosity and thus requires a higher injection pressure to fill the mold. The curve shape of injection pressure versus fill time strongly depends on the material used, as well as on the cavity geometry and mold design. If the required pressure exceeds what can be tolerated by the decoration due to the flow-length/wall thickness ratios, special molding processes that permit low-pressure molding should be considered, as mentioned earlier [56].

For IMD or in-mold lamination, mold-wall temperature control is very important, because the decorative laminate is sensitive to the temperature. The melt temperature would normally exceed the maximum temperature that the decorative laminate can withstand. That is the reason why an additional backing layer is needed, which provides the insulation against the melt while reducing the possibility of melt break-through and folding. Meanwhile, because of the insulating effect of the decorative laminate, the mold-wall temperature on the decorated side should be set lower than the other side to promote balanced cooling and avoid part warpage.

9.5.5   Applicable Materials

As far as the base material is concerned, the processes are feasible with virtually all thermoplastics [57]. The largest volume resin used in in-mold lamination is PP, mostly for automotive applications. A wide range of other materials, including ABS, ABS/PC blends, PS, modified PPO, polyesters, PBT, PA 6, PA 66, and PE, have also been used successfully.

On the other hand, decorative laminate must have good thermal stability and resilience due to their exposure to high injection temperature and pressure. Color change that occurs at the outer surface of the decoration typically results from thermal damage. In addition, the extensibility (stretching ability) of the decoration laminate is crucial for its applicability. Poor extensibility leads to tearing of the decoration laminate, whereas excessive extensibility results in overstretching that causes difference in surface brightness or show-through of the base material. Finally, compatibility and adhesion of the laminate with the base (carrier) resin are important considerations. The adhesion of the laminate to the part can be done through using

a heat-activated adhesive layer or by melt penetration into a fabric backing to form a mechanical bond.

Because the recyclability of the laminated composite is becoming an important issue, the use of a single polyolefin-based system for the decorative, intermediate, and base layers seems to be a viable and cost-effective approach [58]. Moreover, due to the special characteristics of the processes, use of regrind as the base material is feasible. For example, recycled PP has been in favor by the automotive industry. It should be pointed out, however, that the presence of the metallic film in the laminate makes the recycling of decorated components more difficult.

9.5.6   Typical Applications

The IMD and in-mold lamination processes are suitable for many situations by providing integrated, single-step surface decoration and greater design freedom. For example, in-mold decoration has been used to produce rooftops, bumper fascia, and exterior mirror housings, as well as automotive lenses (cf. Fig. 9.20) and body-side molding with mold-in colors to help automakers eliminate costs and environmental concerns associated with painting. On the other hand, in-mold lamination has been used widely for automotive interior panels and other application areas. For example, Figs. 9.21 and 9.22 show the automotive interior panels and a shell chair featuring textile surfaces, respectively. These parts were made using a low-pressure in-mold lamination technique [35]. Finally, Table 9.7 lists some of the typical applications for
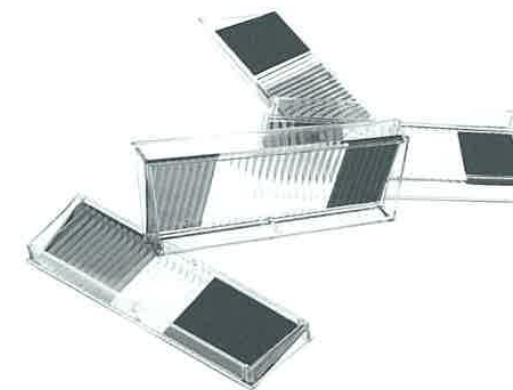

Figure 9.20 Automotive lenses with transparent film made by low-pressure in-mold decoration process. (Courtesy of Hettinga Technologies, Inc., Des Moines, Iowa, USA.)
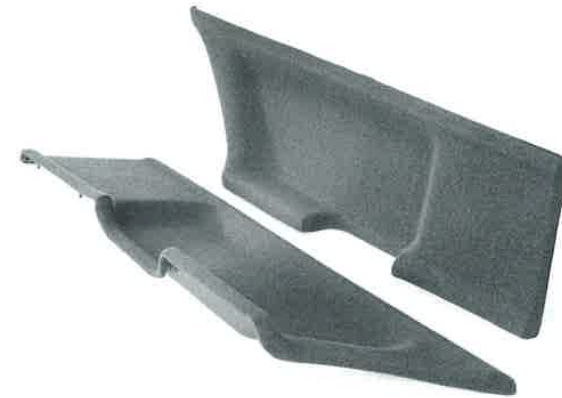
Figure 9.21 Automotive interior panels molded with in-mold lamination using low-pressure injection molding. (Courtesy of Hettinga Technologies, Inc., Des Moines, Iowa, USA.)

Figure 9.22 A shell chair featuring a textile of pattern/strips has been successfully molded with in-mold lamination using a low-pressure injection molding technique called inverted force molding, IFM [35]. (Courtesy of Hettinga Technologies, Inc., Des Moines, Iowa, USA.)

Table 9.7 Typical Applications for In-Mold Decoration and In-Mold Lamination [25,55]

Application area	Molded parts
Automotive industry	Column cladding panels, consoles, door panels, headliners, heater/air conditioning controls, pillars, seat back panel, ventilation grilles, wheel covers, glove boxes, horn knobs, scale for tachometers, emblems
Household appliances	Appliance control panels (e.g., washing machine cover plates, front of microwave oven, toaster housing, etc.)
Telecommunication devices	Key pads and membrane switches for cellular phones
Radio industry	Cassette packs, covers for cassette players, video front plates
Sport equipment	Hockey sticks, water skis
Cosmetic industry	Caps for containers, powder-compact lids

various industrial sectors, which used to be manufactured by a labor-intensive and costly adhesive lamination process.

9.6 Insert and Outsert Molding

Insert molding is a process by which components consisting of metal or other materials are pre-placed in the cavity and then incorporated into a part as it is being molded. On the other hand, in the outsert molding process, the polymer melt is injected onto a metal frame (usually sheet metal) with holes punched out, at which the functional elements formed by the plastics are kept and anchored. For moderate to large production quantity, considerable savings in assembly and finishing can be realized by insert or outsert molding, especially with automated robot or machine-loading/unloading systems for insertion.

9.6.1 Insert Molding Process Description

For some applications, especially those in automotive, electric and electronics, medical, and fiberoptics, it is desirable to join plastics with metal parts. This can be done by the insert or outsert molding process. Insert molding is very similar to multicomponent injection molding (or overmolding, see Sec. 9.11) and in-mold decoration (see Sec. 9.5) in the way that an insert is pre-placed in the mold to be subsequently over-molded by the polymer melt. Note that for multicomponent injection molding, the insert is typically a premolded plastic component, whereas a film, foil, or fabric with desirable printing or color is used for IMD. Due to the similar insert approach, insert molding sometimes has been used to refer to these two processes. This section will focus on insert molding that incorporates inserts other than pre-fabricated plastic component, film, foil, or fabric.

For insert molding, polymer melt is injected into the mold where inserts have been preloaded. These insert components, such as fasteners, pins, rings, studs, terminals, thread inserts, and metallic strips that provide electric conductivity, are to be incorporated into the final molded parts. Figure 9.23 shows two of the typical parts made by insert molding. These inserts need to be loaded in the mold manually or by robot before polymer melt is injected into the cavity. The insert components are fixed in the mold by an undercut and/or by magnetic force [49]. The use of vertical insert molding machines with molds on a rotary table and automation support systems are quite common because they facilitate the loading and positioning of inserts. In addition, it can increase the production rate by concurrently loading the inserts in one mold and injecting the polymer in the other. Automation of insert loading also leads to faster cycle times, improved quality with precision, and enhanced production efficiencies.
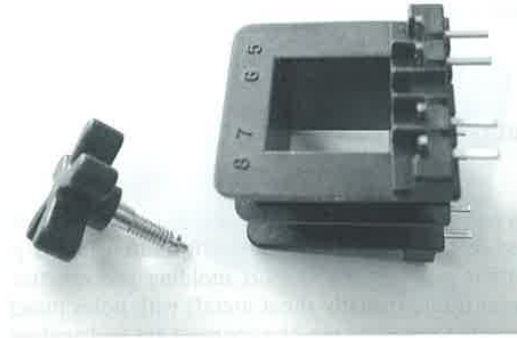
Figure 9.23 A metal thread overmolded with plastic cap for better grip and an electrical coil component with molded-in metallic pins. (Parts courtesy of Engineering Industries, Inc. Madison, and Quantum Devices, Barneveld, Wisconsin, USA.)

One of the prospective developments of insert molding is to compete with transfer molding for encapsulation of integrated circuit (IC) chips. Such a new application is made possible by development of new series of epoxy molding compounds (EMC) that can stand extended residence time inside the barrel unit of an injection molding machine. An article on machine development on insert molding and this new IC packaging application can be found in Ref. [59].

9.6.2   Outsert Molding Process Description

The concept of outsert molding has been developed by exchanging the roles of metals with that of plastics in insert molding. The most common application is to inject polymer melt onto a metal frame (e.g., a sheet metal) preplaced and clamped in the cavity. The metal frame comes with punched-out holes where the plastic anchors and forms multiple functional elements, such as axles, bearings, bosses, guides, pins, spacers, shafts, snap connections, spiral and tongue springs, turning knobs, longitudinal and rotary movable elements. The possibility of having movable components on the sheet metal (e.g., a deflectable spring leaf, a sliding element movable in the longitudinal direction, or rotary turning knobs) result in considerable savings compared with metal components. Such a composite building method combines necessary load-bearing strength from the metal frame with design freedom from the material and processing advantages of thermoplastics. In the meantime, the economic savings may amount up to 40% compared with conventional manufacturing methods, which individual prefabricated elements of plastics or other materials have to fit with the sheet metal [60].
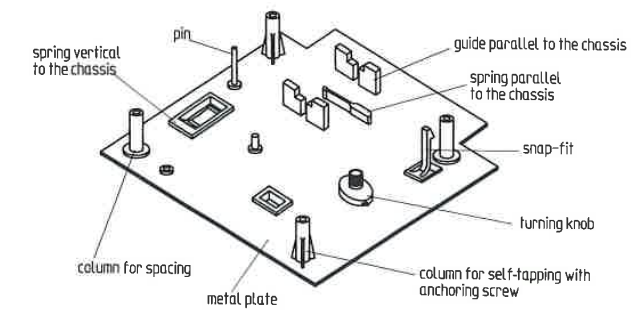
Figure 9.24 Various elements fabricated by outsert molding. (From *Injection Molding—An Introduction* (1995), Pötsch G. and Michaeli, W., Hanser, Munich, p. 181, Fig. 8.14.)

In outsert molding, the plastic elements to be molded onto the metal frame are connected by a distributor spider with restricted gating or, if the distance is too small, by channels so that they can be molded with a single shot. These channels are either detached after removal of the article from the mold or remain on the metal frame, provided that they do not interfere with the product functions. Due to a larger thermal expansion coefficient of the plastics as compared with that of metals, care should be taken so that the shrinkage of these connecting channels does not deform the metal frame or cause the plastic elements to break. With such a multicavity system, dozens of functional elements can be incorporated on the metal frame base. As an illustration, Figs. 9.24 and 9.25 show the various elements fabricated by outsert
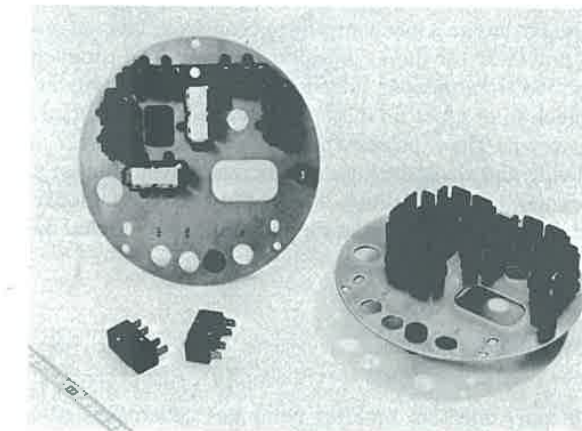

Figure 9.25 Building elements of a tube tester made with outsert molding [60].

molding and the building elements of a tube tester made with outsert molding, respectively.

In terms of the metal frame base, the sheet metals are usually 1- to 2-mm thick with thickness tolerance being smaller than 0.1 mm. Burrs at the edges of punched holes should be removed. Prior to injection molding, the sheet metal must be degreased and flattened. Two drilling holes with a distance larger-than-half of the sheet metal dimension are used for accurate positioning of the sheet metal inside the cavity. Expansion of the sheet metal should be taken into account as the metal touches the hot mold. It is recommended that one drilling hole be carried out as an oblong hole.

9.7   Lamellar (Microlayer) Injection Molding

Lamellar (microlayer) injection molding uses a feed-block and layer multipliers that combine melt streams from dual injection cylinders to produce shot volume in distinct microlayer morphological structure to be directly injection molded. The resulting parts that combine different resins in a layered structure exhibit enhanced properties, such as the gas barrier property, dimensional stability, heat resistance, and optical clarity, compared with parts made of single-phase resin or conventional polymer blends.

9.7.1   Process Description

Lamellar injection molding technology has been available since early 1990s [61]. It has been developed to create a mechanically micron-scale, layered morphology blend of two or more polymers for direct injection molding. The production of multimicrolayers is done via simultaneous injection of two components at predetermined rates in a feed-block to produce an initial, sandwiched layer structure (i.e., A–B–A). This resultant streams then passes through a series of layer multipliers that repeatedly subdivide and stack the layers to increase layer number and reduce individual layer thickness (cf. Figs. 9.26–9.28). At the end, the lamellar morphology is mechanically retained and remained in the final injection molded parts, as shown in Fig. 9.29 [62].

9.7.2   Process Advantages

Lamellar (microlayer) injection molding combines advantages of coextrusion and injection molding by coupling simultaneous injection with layer multiplication to produce a finely subdivided microlayer melt stream into complex multicavity molds
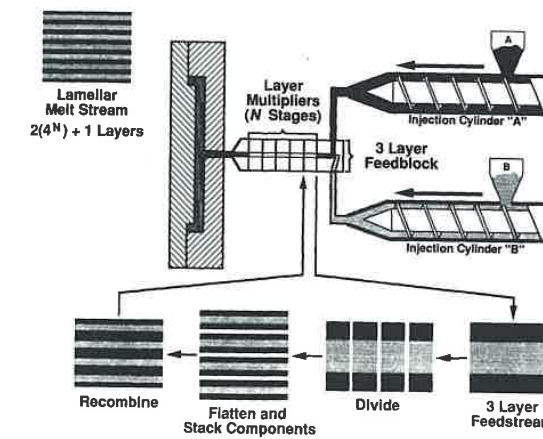
Figure 9.26 Schematic of lamellar injection molding showing the creation of micron-scale, layered morphology blend through the layer multipliers (see Figs. 9.27 and 9.28). (From *Innovation in Polymer Processing—Molding*, Stevenson, J. F., (Ed.) (1996), Hanser, Munich, p. 297, Fig. 8.2.)
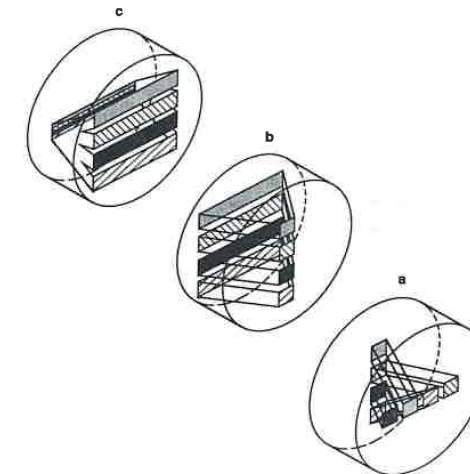

Figure 9.27 Three-piece layer multiplier assembly: (a) subdivision and layer rearrangement, (b) spreading of individual substreams to reestablish the original width, (c) recombination of individual substreams in a contraction zone that reestablishes the original cross-section. (From *Innovation in Polymer Processing—Molding*, Stevenson, J. F., (Ed.) (1996), Hanser, Munich, p. 300, Fig. 8.4.)
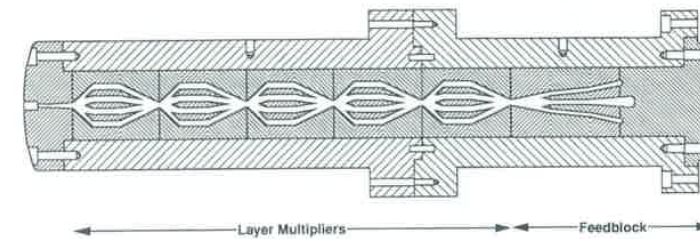
Figure 9.28 Three-layer feed-block and layer multiplication assembly contained in a tubular sleeve. (From *Innovation in Polymer Processing—Molding*, Stevenson, J. F., (Ed.) (1996), Hanser, Munich, p. 300, Fig. 8.5.)

without being restricted to circularly symmetric parts or metering individual mold cavities. In terms of producing continuous lamellar structure, it provides a superior alternative to conventional polymer melt blending technique for which the resulting morphology strongly depends on the material properties of the constituent materials and the processing parameters. It has been reported that one or two orders of magnitude reductions in oxygen permeability can be realized with lamellar injection molding relative to monolayer PET and conventional blends [62]. In addition, a 200-fold reduction in hydrocarbon permeability with HDPE-polyamide (nylon) 6 microlayer structure compared with HDPE has been achieved. Furthermore, significant enhancement in chemical and temperature resistance, dimensional stability, and optical clarity were also made possible through lamellar injection molding.
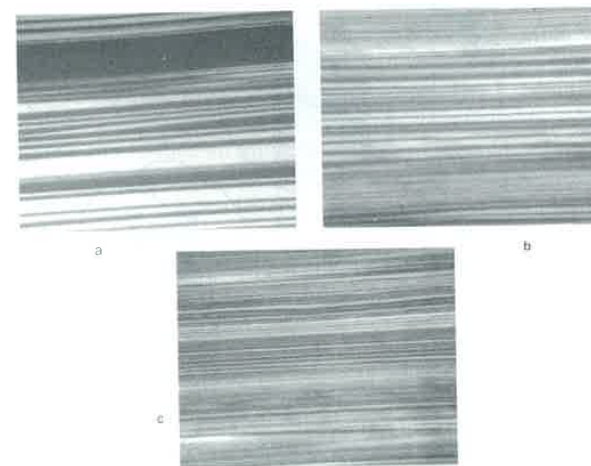
Figure 9.29 Micrograph of multilayer cross-sectional structure produced by the lamellar injection molding process. (From *Innovation in Polymer Processing—Molding*, Stevenson, J. F., (Ed.) (1996), Hanser, Munich, p. 306, Fig. 8.12.)

9.7.3 Process Disadvantages

The disadvantages of lamellar injection molding are the initial equipment cost and learning curve to acquire process know-how, which are typical of adopting any new technology. Major challenges of this process include determination and control of proper lamellar structure for the end application, effects of fountain flow behavior at the melt front (cf. Chap. 3), and high shear rate during injection molding on the break up and coalescence of lamellar structure, adhesion of incompatible polymers, and recycling of process scraps and postconsumer parts.

9.7.4 Applicable Materials

Lamellar injection molding has been employed with diverse combinations of polymers, especially with multiphase systems that combine commodity or engineering resins with a small quantity (2 to 20%) of barrier resin. Mutually reinforced composite properties can also be attained with proper selection of materials and morphology. Potential materials for lamellar injection molding applications are listed in Table 9.8.

Table 9.8 Potential Materials for Lamellar Injection Molding Applications [62]

Material combinations	Target benefits
PC-PET	Clarity, barrier, and chemical resistance
PC-PBT	Temperature and solvent resistance
Polyolefin-adhesive-EVOH	Gas and solvent barrier
PET-PEN	Barrier and temperature resistance and clarity
Polyolefin-adhesive-polyamide	Gas and solvent barrier
Transparent impact modified polystyrene-polyamide	Gas barrier and clarity
ABS-thermoplastic polyurethane	Solvent resistance
PC-thermoplastic polyurethane	Solvent resistance, temperature resistance, and optical clarity
Combinations of filled and unfilled polymers (e.g., PP-filled PP, PC-filled PC, PC-filled ABS, ABS-filled PC)	Preferred balance of properties, coefficient of linear thermal expansion, toughness, and temperature resistance
Combinations of brittle and ductile polymers	Mutually reinforced mechanical properties
Combinations of engineering thermoplastics with thermotropic liquid crystalline polymers (T-LCP)	Reinforcement with T-LCP
Brominated PC copolymers with ABS	Ignition resistance
Combinations of virgin and post-consumer recycled resins	Material property modification and cost savings

9.7.5   Typical Applications

Lamellar injection molding is believed to be advantageous over other competing processes in diverse application areas. In particular, potential application areas for lamellar injection molding technology include: (1) structural parts that require temperature resistance, chemical resistance, and/or dimensional stability, (2) houseware and durable goods that require optical clarity and/or temperature and solvent resistance, (3) health-care devices and components, (4) food and chemical packaging products that require barrier resistance, and (5) automotive components such as fuel and hydraulic systems [62].

9.8   Low-Pressure Injection Molding

Low-pressure injection molding (LPIM) is essentially an optimized extension of conventional injection molding. This process differs from traditional (high-pressure) injection molding in that it integrates a series of innovative practices that keep the injection pressure (and, consequentially, the clamp force) at the lower limits. The major benefits of LPIM include significant reduction of the clamp force tonnage requirement, less-expensive molds and presses, and lower molded-in stress in the molded parts. It also facilitates reduction of manufacturing cost by incorporating decorative film and/or textile into the molded parts.

9.8.1   Process Description

LPIM can be achieved by properly programming the screw rotation speed and the plasticating back pressure to control the melt temperature profile of the shot volume as well as by precisely profiling the injection speed and pressure to maintain a generally slow and controlled melt front velocity inside the tool. It also uses a generous gate size, a novel type of film gate, and/or valve gates that open and close sequentially based on volume of material injected to reduce the flow length while eliminating the weld lines. The packing stage is eliminated after complete injection of an exact shot volume with no shot cushion. These practices have been implemented into a number of processes by Hettinga [35]. Notable LPIM processes from these innovations are the Inverted Force Molding (IFM®), which maintains a constant melt front speed during injection, and the low-stress Thermoplastic Solid Molding (TSM™) for

molding solid thermoplastics or molding with delicate films and textiles. By employing advanced electronics and hydraulics married with sophisticated software control, the process can maintain a solid, fully plasticized melt front with low pressure. The following sub-sections describe the characteristics of low-pressure injection molding process.

Control of Injection Temperature Profile

In LPIM, it has been indicated that nearly all of the heat required for mold filling comes from plastication as a result of the gentle injection speed and generous gate size [63]. It is different from conventional injection molding for which a substantial portion of the heat is generated by viscous heating. The control of shot volume temperature profile is done by manipulating the screw rotation speed (RPM) and the back pressure at the screw tip during the plastication stage. Note that the effective L/D ratio of the reciprocating screw decreases (by a length of generally 1 to 5 diameters) as the screw retracts in preparation for the shot volume. To compensate for the changing plastication length of the screw as it rotates and moves backward, an electrohydraulic device is employed to generally increase the back pressure at the screw tips to enhance the mixing and shearing (cf. Fig. 9.30). The screw RPM speed also follows a ramps-up-and-down profile to work with the back pressure for building up the desirable temperature profile within the shot volume (cf. Fig. 9.30).

Control of Injection Pressure and Injection Rate

The injection pressure profile is set in such a way that it delivers a controlled injection speed during filling. As illustrated in Fig. 9.30, the injection rate starts slowly to assure an even flow into the mold. Once the melt enters the cavity, the rate is set so that the melt front travels at the same speed throughout the injection stage. The injection pressure profile is generally in the shape of an inverted U (cf. Fig. 9.30), which
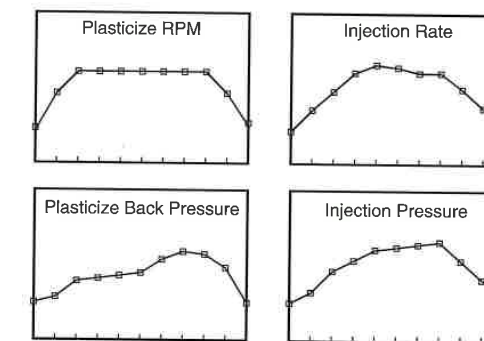

Figure 9.30 Typical profiles for screw RPM speed, back pressure, injection rate, and injection pressure for low-pressure injection molding.

reduces the rapid build up in the clamp force at the end of the cycle commonly seen in conventional injection molding [64].

Because the material solidifies almost immediately as it comes into contact with the cold mold wall, the velocity at the melt front determines the flow-induced stress and the degree of molecular and fiber orientation in regions adjacent to the part surface. Variable orientation within the part, as a result of changing velocity at the melt front during filling, leads to differential shrinkage and, thus, part warpage. It is desirable, therefore, to maintain a constant velocity at the melt front to generate uniform molecular and fiber orientation throughout the part.

Accurate Shot Size Control with No Cushion and Packing Stage

LPIM does not employ distinct high-pressure packing and holding stages as does conventional injection molding. The exact shot size with no cushion is injected into the mold in a carefully controlled, even fashion. Setting the correct injection volume is nevertheless important because a very minimum of packing is necessary for filling the cavity safely. Because the gate does not freeze off after the mold cavity fills completely, some compensation for thermal shrinkage is possible so there is no need for high-pressure packing and holding. It is conceivable that voids or sink marks will occur at thicker sections, ribs, and the like, due to the fact that compensation for the polymer volumetric shrinkage is not complete. For molding with approximately constant wall thickness, however, LPIM is thoroughly suitable [57]. In addition, it is claimed that the LPIM generally allows material to be molded at lower temperature. This results in comparable cycle time, even if the cavity filling is slower.

Multistation Injection Molding Machine

The lower pressure and clamp force requirements permit the use of less expensive tools. Because the process requires only one quarter to one third as much clamp tonnage for molding most parts, aluminum rather than steel can be used for tooling. In addition, the control hardware and software allow a single injector to serve multiple mold stations with different parts of various size, shape, weight, and configuration. As an example, the six-station, LPIM machine shown in Fig. 9.31 permits different parts to be molded consecutively without retuning the machine.

Multi-Point Volumetric Injection Control™

Multi-Point Volumetric Injection Control™ optimizes mold filing of large parts and highly viscous materials by sequentially opening and closing valve gates at strategically selected locations. The timing of gate opening and closing is controlled by volume injected instead of using standard time-based controls as the other cascading (or sequential valve-gate control) injection. The advantage of timing-by-volume approach is that the switching process of the hot runners are always connected to the current stage of filling, regardless of the speed changes required by the process. As an illustration example, Fig. 9.32 shows the mold filling in a bumper fascia. At first,
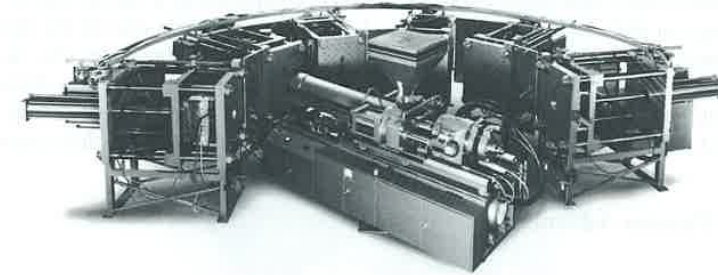
Figure 9.31 Multistation injection molding machine. (From *Innovation in Polymer Processing—Molding*, Stevenson, J. F., (Ed.) (1996) Hanser Publishers, Munich, p. 204, Fig. 5.7. Courtesy of Hettinga Technologies, Inc., Des Moines, Iowa, USA.)

resin is injected through the center gate. After the resin flows past the two downstream gates, it triggers the opening of those two gates, followed by closing of the first gate shortly. The process continues until the entire cavity is filled. It is conceivable that the use of multiple gates reduces the flow length, thereby, reducing the injection pressure requirement [65]. In addition, the sequential opening and closing of the gates eliminate the formation of weld lines. If necessary, the melt injection speed profile can be adjusted from gate to gate. For situations where packing pressure has to be used, all the valve gates can be opened again at the end of the injection stage.

Laminate Molding with Textiles, Foils, or Films

In-mold lamination or simply laminate molding is also known as composite or back molding. This process involves molding of plastics over layers of textile, film, or other materials preplaced inside the mold to form a laminated structure (see Sec. 9.5 for detailed descriptions on in-mold lamination). The inclusion of textile or film provides
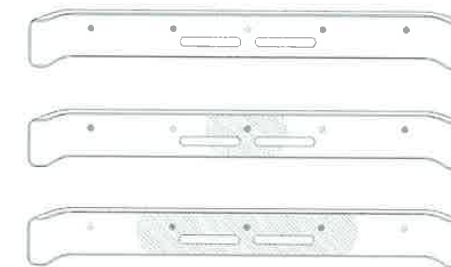

Figure 9.32 A bumper fascia being filled by the Multi-Point Volumetric Injection Control™ or MPVIC Control. (Courtesy of Hettinga Technologies, Inc., Des Moines, Iowa, USA.)

an insulation layer that allows plastics melt to be injected at a slower rate to avoid the damage or shift of the textile or film. Laminate molding is one of the most successful applications with low-pressure molding because it eliminates the need for multiple-layer fabrics and special glue backings on textiles. A study has shown that the one-step laminate process offers weight reduction (12%) and cost saving (64%) compared with the traditional method of gluing the fabric to the molded part [63].

9.8.2 Process Advantages

The LPIM generally offers the following process advantages:

- Reduced tonnage requirements and thus lower cost molds
- Lower stresses in the molded parts and improved material properties
- Molded-in fabric or film capabilities
- Large part capability and material savings of 5 to 8%
- Lower melt temperatures and shorter overall cycle times
- Lower gate and cavity pressures
- Reduced tooling costs and overall manufacturing costs

9.8.3 Process Disadvantages

Because the various practices and techniques employed by low-pressure injection molding favorably reduces the injection pressure requirement, this process does not introduce any adverse process disadvantages for its intended applications.

9.8.4 Applicable Materials

LPIM is well suited to processing a broad range of materials, such as thermoplastics, thermosets, polymer alloys and blends, filled materials, recycled thermoplastics, and even rubber. In particular, polypropylene is used extensively with the LPIM due to the lower cost and improved physical and mechanical properties. There are a broad range of compatible textiles and films that can be molded with polypropylene using in-mold lamination. For fiber-filled materials, the generous gate size reduces the possibility of fiber breakage.

9.8.5 Typical Applications

In addition to the in-mold lamination components (cf. Figs. 9.21 and 9.22), the application of LPIM includes a variety of automotive components ranging from dashboards, consoles, interior panels (cf. Fig. 9.33), kick panels, and visors to glove boxes, lenses (cf. Fig. 9.20), headliners, bumpers, and trims. Other nonautomotive applica-

Figure 9.33 Automotive interior panels that eliminate heavy backed, expensive textile. (Courtesy of Hettinga Technologies, Inc., Des Moines, Iowa, USA.)

tions are material-handling containers (cf. Fig. 9.34), shipping pallets, business machine housings (cf. Fig. 9.35), lavatories (cf. Fig. 9.36), and building panels. Potential large applications include truck cabs and boat hulls.

9.9 Microinjection Molding

Microinjection molding (also called micromolding) produces parts that have overall dimension, functional features, or tolerance requirements that are expressed in terms of milli- or even micrometers. Due to the miniature characteristics of the molded parts, it requires a special molding machine and auxiliary equipment to perform tasks such as shot volume control, evacuation of mold (vacuum), injection, ejection, inspection, separation, handling, deposition, orientation, and packaging of molded parts. Special techniques are also being used to make the mold inserts and cavities.

9.9.1 Process Description

The demand for miniature injection molded parts and the equipment and processing capability to produce them with desirable precision began around 1985 and has been

Figure 9.34 An assortment of thin-walled kitchen-wares molded from high-density linear poly-ethylene, HDLPE. Each part was produced using the Inverted Force Molding (IFM) process on a 275-ton press. (Courtesy of Hettinga Technologies, Inc., Des Moines, Iowa, USA.)

growing ever since [66]. Among the various micromolding processes (see, e.g., [67,68]), injection molding possesses the advantages of having a wealth of experiences available in conventional plastics technology, standardized process sequences, and a high level of automation and short cycle times [69].

Figure 9.35 Business machine housings molded with Thermoplastics Cellular Molding, (TCM®) process. (Courtesy of Hettinga Technologies, Inc., Des Moines, Iowa, USA.)

Figure 9.36 Lavatory sink molded with Inverted Force Molding (IFM), which permits high gloss decorative films to be molded across the entire surface without damage or distortion of color on deep draws of film. (Courtesy of Hettinga Technologies, Inc., Des Moines, Iowa, USA.)

Microinjection Molding Classifications

Although there is no clear way to define microinjection molding, applications of this process can be broadly categorized into three types of products or components [70]:

- Microinjection molded parts (micromolding) that weigh a few milligrams to a fraction of a gram, and possibly have dimensions on the micrometer (μm) scale [e.g., micro-gearwheels, micro operating pins (cf. Fig. 9.37)]
- Injection molded parts of conventional size but exhibit microstructured regions or functional features (e.g., compact disc with data pits, optical lenses with micro-surface features, and wafer for making microgearwheels with Plastic Wafer Technology) [72] (cf. Fig. 9.38)
- Microprecision parts that can have any dimensions, but have tolerances on the micrometer scale (e.g., connectors for optical fiber technology)

Machine Requirements

Due to the previously mentioned characteristics, these applications all require special attention to the machine equipment, processing, making of molds, and the like.

Figure 9.37 An microinjection molded operating pin by HB-Plastic GmbH, Korneuburg, Austria [71].

Although modern conventional injection molding machines can achieve impressive results, these machines must be adapted to meet the special requirements of microinjection molded parts such as [69]:

- Small plasticating units with screw diameters ranging from 12 to 18 mm and a shorter screw length with L/D ratio around 15 to avoid material degradation from prolonged residence (dwell) time
- Precise shot volume control and desirable injection rate
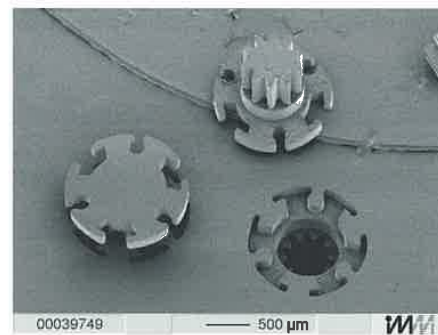
Figure 9.38 Microgearwheels manufactured by means of Plastic Wafer Technology [72].

- Repeatable control such as fill to pack switchover based on screw position or cavity pressure (preferred)
- Capability to raise the mold-wall temperature to such a level (sometimes slightly above the melting temperature of the polymer) to avoid premature solidification of ultrathin sections
- Mold evacuation if the wall thickness of the microinjection molded part is down to $5\,\mu m$, which is the same order as the dimension of vent for air to escape
- Shutoff nozzle to avoid drooling from the nozzle due to the high processing temperature of the melt
- Precise alignment and gentle mold-closing and -opening speeds to avoid deformation of delicate microinjection molded parts
- Special handling technique to remove and molded parts for inspection and packaging
- Possible local clean room enclosure or laminar flow boxes to avoid contamination of microinjection molded parts

It is apparent that with diminishing part volume and shot size, the conventional injection molding machine is no longer an economically viable solution for microinjection molding applications. As a result, many "micromolding machines" have been developed and have become commercially available (see, e.g., [66]). In addition to the various features described earlier, these machines are sometimes equipped with separate metering and injection ram (pistons) and screw design. These features are targeted to meter the shot volume accurately and to eliminate problems associated with material degradation resulting from by-pass and dead corners with the conventional injection screw. Because the size and weight of the microinjection molded parts differ significantly from the conventional parts, certain steps have to be taken to ensure proper part ejection. For example, a vision system can be installed in the molding machine to confirm ejection of the miniature parts. In addition, part removal can be performed by suction pads, which keep parts separate and oriented for quality control and packaging, or by electrostatic charging or blowing out. Traditional quality control methods, such as measuring the part weight, becomes impractical with microinjection molding. New quality control techniques employ a video inspection system for sorting acceptable and unacceptable parts. Because microinjection molded parts are usually part of the assembly, they are packaged in an oriented way ready for assembly.

Mold Making (Tooling)

Traditional methods of tooling, such as various machining methods and electrical discharge machining (EDM), can be used for making the mold for microinjection molded parts; however, they have quickly reached their limitations with decreasing dimensions of mold inserts and cavities. Existing technologies in the field of microelectronics have been employed to fabricate mold inserts and cavities for microinjection molding. One common practice is the LIGA technique, which is an acronym derived from the German words for deep x-ray lithography, electroforming

(electroplating), and injection molding replication [73,74]. Other processes include microcutting, ultraprecision machining, laser machining, and micro-EDM technologies [69].

Molding Process

Microinjection molded parts typically have features on the order of a few micrometers to millimeters, and aspect ratios between 1 and 100. Several requirements are needed in order to reproduce these features with high fidelity. To be specific, the mold needs a special heating and cooling system. A few systems have been developed to provide the desirable dynamic mold temperature control. For example, the "variotherm" process control employs two oil circuits at different temperatures that heat up and cool the mold at filling and cooling stages, respectively [75]. On the other hand, the induction heating technique generates a peak mold temperature using an inductive system prior to injection [76]. Moreover, satisfactory results have been reported using electric cartridge heaters to control the mold temperature [77]. The use of adapted mold sensors, high-precision mold guides, mold evacuation system, integrated runner pickers and part removal robots, automatic gate-cutting systems, and mold cleaning systems that activate during each cycle are all vital to controlling the manufacturing processes and efficient handling and packaging of microinjection molded parts.

To ensure proper cavity filling, high injection speeds and pressures, maximum permitted melt temperatures, and elevated mold-wall temperature controls are needed. Large runner and sprues are used to have a shot size large enough to control the process and switchover reliably while moving polymer melt through the machine system to avoid degrading the material.

9.9.2   Process Advantages

The microinjection molding is intended for producing miniature parts. It does not compete with other macroinjection molding processes.

9.9.3   Process Disadvantages

Because ultrasmall parts do not weigh enough to register on a machine control system, some system uses oversized runners so that the machine can accurately set and monitor part production. Under these conditions, the runners can constitute up to 90% of the total shot weight. Waste is considerable because the material in the runners cannot be recycled most of the time for microinjection molding application. Finally, due to the typically high surface-to-volume ratios of the parts, the mold has to be heated to temperatures greater than the melting temperature during injection to prevent premature solidification, which leads to prolonged cycle time.

9.9.4   Applicable Materials

Almost any material suitable for macroscopic molding can be micro-molded. Materials reported for microinjection molding include POM, polycarbonate (PC), polymethylmethacrylate (PMMA), PA, liquid crystalline polymers (LCP), polyetherimide, and silicone rubber. Reaction injection molding has also been applied using material on the basis of acrylate, amides, and silicones [67].

9.9.5   Typical Applications

There is a rapidly growing demand for microinjection molded parts in such sectors as optical telecommunication, computer data storage, medical technology, biotechnology, sensors and actuators, microoptics, electronics, and consumer products, as well as equipment making and mechanical engineering. Examples of microinjection molded parts include watch and camera components, automotive crash, acceleration, and distance sensors, read–write heads of hard discs and CD drives, medical sensors, micropumps, small bobbins, high-precision gears, pulley, and helixes, optical fiber switches and connectors, micromotors, surgical instruments, and telecommunication components [66,78].

9.10   Microcellular Molding

Microcellular molding (also commercially known as MuCell process) blends "supercritical" gas (usually nitrogen or carbon dioxide) with polymer melt in the machine barrel to create a single-phase solution. During the molding process, the gas forms highly uniform microscale cells (bubbles) of 0.1 to $10\,\mu m$ in diameter, and the internal pressure arising from the foaming eliminates the need of packing pressure while improving the dimensional stability. The original rationale is to reduce the amount of plastic used without sacrificing the mechanical properties. Because the gas fills the interstitial sites between polymer molecules, it effectively reduces the viscosity and glass transition temperature of polymer melt; therefore, the part can be injection molded with lower temperatures and pressures, which leads to significant reduction of clamp tonnage requirement and cycle time.

9.10.1   Process Description

Microcellular plastics (MCPs) are single-phase, polymer-gas solutions by dissolving or saturating a polymer in a supercritical fluid (CO_2 or N_2) and then precipitating it by triggering polymer nucleation by adjusting process conditions, such as pressure
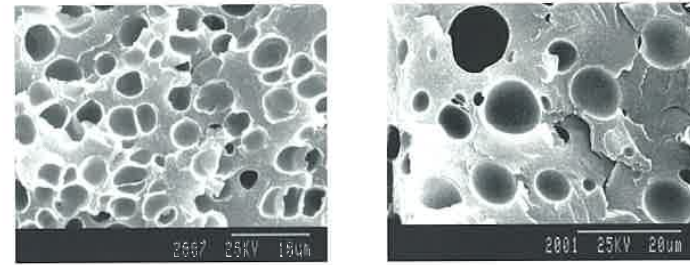
Figure 9.39 Micrographs of microcellular polystyrenes prepared under different conditions. (Photos courtesy of Institute of Chemistry, The Chinese Academy of Science.)

and temperature. The resulting MCPs are plastic foams with cell diameter sizes from 0.1 to 10 μm (cf. Fig. 9.39) and cell densities from 10^9 to 10^{15} cells per cubic centimeter. MCPs were initially conceived by Suh and co-workers at MIT as a means of reducing material consumption in mass-produced plastic parts [79]. The underlying rationale is to create enough voids smaller than the pre-existing flaw in polymers so that the amount of plastic used could be reduced without compromising the mechanical properties. Even though realizing a part weight reduction of 5 to 95% by replacing plastics with gas, the microcells also serve as crack arrestors by blunting crack tips, thereby, greatly enhancing part toughness [79]. When properly prepared, microcellular polystyrene (PS) has five times the impact strength of its unfoamed counterpart [80,81]. The fatigue life of microcellular polycarbonate (PC) with a relative foam density of 0.97 is four times that of its solid counterpart [82]. Furthermore, because the gas fills the interstitial sites between polymer molecules, it effectively reduces the viscosity [83,84] and the glass transition temperature of the polymer melt [79,85–87]. The material, therefore, can be processed at much lower pressures and temperatures. Table 9.9 lists the comparison of some of the properties between microcellular plastics and its solid unfoamed counterpart.

In addition to typical molding procedures, microcellular molding process involves four distinctive steps:

- Gas dissolution—atmospheric gas (N_2 and CO_2) is injected into the machine barrel to form a polymer-gas solution for processing.
- Nucleation—a large number of nucleation sites (orders of magnitude higher than conventional foaming processes) are formed by rapid and substantial pressure drop.
- Cell growth—cell growth is controlled by the processing conditions such as melt pressure and temperature.
- Shaping—the shaping of the part takes place inside the mold.

Although a number of fluids (or gases) can be dissolved into the polymer to form MCPs, many of them are either corrosive, less cost effective, or hazardous. Two of common gases used in microcellular molding process are N_2 and CO_2. The solubility of CO_2 is higher than that of N_2; however, N_2 tends to provide finer cell structure and gives better surface finish. To enhance the diffusion rate of N_2 and CO_2 into polymer, mechanical blending and shearing of the gas with the polymer is used to increase the contact surface area and reduce the striation thickness. In addition, the solubility of gas is further improved by bringing it to a "supercritical" state. At this state (the so-called supercritical fluid, SCF, state), gas behaves like a fluid so that it can be metered precisely into polymer melt, and yet it has the high diffusion rate of a gas for even distribution and dissolving in the melt. The introduction of supercritical gas into polymer melt is provided by a free-standing SCF conditioning system, which uses an air pump to bring the gas to the supercritical fluid state. The fluidlike gas is metered into the injection barrel by a solenoid metering valve, which is controlled by the machine controller. A specially designed screw is used to create the gas-polymer solution for both the microcellular molding process as well as for the solid plastics injection molding.

The solubility of gas increases with increasing pressure and decreasing temperature. Prior to injection, the melt is kept under sufficient pressure with higher screw back pressure to keep the gas in solution. In addition, the process requires a certain injection speed to keep the gas in solution before gas foaming starts in the mold [88]. Packing is not desirable because it impedes the foaming of dissolved gas. Furthermore, the internal pressure of growing cells takes the place of the holding pressure.

During injection molding, the formation of microcells depends on the processing conditions such as melt pressure and temperature. The size of the cells is generally inversely proportional to the cell density, both of which are determined by the thermodynamics of cell nucleation and growth and the amount of gas dissolved in the

Table 9.9   Property Comparisons of Microcellular Plastics with Unfoamed Solid Plastics

Properties	Unformed Plastics	Microcellular Plastics (MCPs)
Specific density ratio ($\rho/\rho_{unfoamed}$)	1.0	0.05–0.95
Glass transition temperature Tg (°C)	100.0	35 (PS)*1
	145.0	65 (PC/CO$_2$)*2
	90.7	50.8 (PMMA/CO$_2$)*3
Viscosity ratio ($\eta/\eta_{unfoamed}$)	1.0	~0.20*4
		~0.01–0.001*5
Tensile strength ratio (T/T$_{unfoamed}$)	1.0	~1.7 (HDPE/iPP/CO$_2$)*6
Elongation ratio at peak (E/E$_{unfoamed}$)	1.0	~1.1 (HDPE/iPP/CO$_2$)*6
Impact strength ratio (I/I$_{unfoamed}$)	1.0	~5.0*7
Fatigue life ratio (F/F$_{unfoamed}$)	1.0	~4.0 (PC/CO$_2$)*8

*1 PS with 10% CO$_2$ (Ref. [85]) *2 CO$_2$/PC at 9 Mpa (Ref. [86]) *3 CO$_2$/PMMA at 15.2 Mpa (Ref. [87]) *4 PS with 4% CO$_2$ (Ref. [84]) *5 Under suitable conditions (Ref. [83]) *6 Ref. [80] *7 Ref. [80,81] *8 At a specific density of 0.97 (Ref. [82]).

polymer. In principle, the larger the amount of gas dissolved in a polymer, the greater the cell density and the smaller of the cell size. The typical cell diameter ranges from 0.1 to 10 μm (as opposed to 250 μm or more with the conventional structural foam molding process). The cell size is so small that the cells cannot be seen by the naked eye; therefore, the molded parts can resemble solid plastics with acceptable physical appearance. The major difference between MCPs and conventional structural (macrocellular) foamed plastics lie in the method to create the formed structure, the size and distribution of the cells, the part-wall thickness limitation, and the physical properties and appearance of the final products. The microcellular molding process uses gas in its supercritical state instead of chemical blowing agents to create evenly distributed and uniformly sized microscopic cells throughout thermoplastic polymers.

9.10.2   Process Advantages

The microcellular molding process is claimed to offer the following processing improvements including potential cost savings [89]:

- Controlled product weight reduction of up to 95% depending on the material used
- Use of SCFs as blowing agents lowers the viscosity of the melt, which results in a substantial decrease in processing temperatures (as much as 78°C), 30 to 50% reduction in hydraulic injection pressure, and 30% or more reduction in clamp tonnage
- Lower melt viscosity and clamp tonnage permit molders to increase the number of cavities or mold larger parts without increasing clamp tonnage
- Faster cycle time due to reduced melt temperature (with lower glass transition temperature), improved mold filling (with lower viscosity), the endothermic reaction of cell nucleation and growth that accelerates cooling, recovery of material's normal glass transition temperature and quick nitrification after gas diffuses out from the MCPs, and the reduction in or elimination of holding pressure and time
- Elimination of sink marks, warpage, and molded-in stresses
- Significant retention of desired mechanical properties, such as high tensile strength, low permanent set, and impact strength, despite a sizable reduction in material content
- Ability of producing parts with thickness as small as 0.5 mm

In addition, molders can run both the microcellular molding process and conventional injection molding on microcellular-molding-capable machines. The microcellular molding process does not require additional chemical blowing (CBAs), hydrocarbon-based physical blowing agents, reactive components, or nucleating agents.

9.10.3   Process Disadvantages

Despite these developments and emerging applications, major challenges in processing MCPs lie in continuous generation of MCPs at an acceptable rate for mass production and control of the state of thermodynamic instability (via temperature and pressure variation) to create fine and uniform microcells throughout the part. In addition, the process requires changes in the machinery components (about 10 to 15% of the machine cost) and licenses. Due to the presence of microcells, it may have limited applications with parts that require clarity. In addition, swirled part surface has been reported that affects the cosmetic appearance. Finally, parts molded by the microcellular molding process needs to be stabilized for the gas inside the cells to diffuse out and its pressure equalize with the atmospheric pressure.

9.10.4   Applicable Materials

Most thermoplastics are suitable for the microcellular molding process, including amorphous and crystalline resins. To date, filled and unfilled materials such as polyamide (PA or nylon), PC, polypropylene (PP), PS, POM, polyethylene (PE), polycarbonate/acrylonitrile butadiene styrene (PC/ABS), high-temperature sulfones, polyetherimides (PEI), as well as polybutylene terephthalates (PBT), Polyetheretherketone (PEEK), and thermoplastic elastomers (TPEs) have been molded.

9.10.5   Typical Applications

For injection molding applications, MCPs will have potential in housing and construction, sporting goods, automotive, electrical and electronic products, electronic encapsulation, and chemical and biochemical applications [79]. Microcellular molding also has a potential in in-mold decoration and lamination due to its low pressure and temperature characteristics that reduce the damage to the film or fabric overlay during molding. Specific application examples include fuse boxes, medical handles, air intake manifold gaskets, automotive trims, and plastic housings.

9.11   Multicomponent Injection Molding (Overmolding)

Multicomponent injection molding (also called *overmolding*) is a versatile and increasingly popular injection molding process that provides increased design flexibility for making multicolor or multifunctional products at reduced cost. By adopting multiple mold designs and shot transfer strategy, this process consists of injecting

a polymer over another molded plastic insert to marry the best features of different materials while reducing or eliminating postmolding assembly, bonding, or welding operations. Special machine equipment with multiple injection units, a rotating mold base, and/or moving cores is available to complete the overmolding in one cycle.

9.11.1 Process Description

Multicomponent injection molding is a special process used in the plastic processing industry, and is synonymous with overmolding, multishot injection molding, and the "in-mold assembly" process. This process, however, is different from coinjection molding, which also incorporates two different materials into a single molded part. To be specific, coinjection molding involves sequential and/or concurrent injection of two dissimilar but compatible materials into a cavity to produce parts that have a sandwich structure, with the core material embedded between the layers of the skin material (cf. Sec. 9.1). On the other hand, for multicomponent injection molding, different polymer melts are injected at different stages of the process using different cavities or cavity geometry. In particular, a plastic insert is first molded and then transferred to a different cavity to be overmolded by the second polymer filling inside a cavity defined by the surfaces of the insert and the tool. The adhesion between the two different materials can be mechanical bonding, thermal bonding, or chemical bonding [90].

Multicomponent injection molding dated back several decades ago as the multicolor molding of typewriter keys to produce permanent characters on the keys. This process has since advanced to allow consistent, cost-effective production of multicolor or multifunctional products in a variety of innovative and commonly used methods. The decision of choosing an appropriate molding technique depends on the production volume, quality requirements, and the molder's capabilities and preference. For example, without any additional equipment investment, one can use two separate molds and the conventional injection molding machine for producing multicomponent injection molded parts. In this approach, the insert is first molded and then transferred to a second mold, where it is overmolded with a second polymer. The disadvantage of this approach, however, is that it involves additional steps to transfer and load the prefabricated insert into the second mold. Nevertheless, the loading and pick up of the insert and molded system can be accelerated and precisely controlled by using robots or automated machine systems.

Another commonly used method of multicomponent injection molding employs a rotating mold and multiple injection units, as shown in Fig. 9.40 [90]. Once the insert is molded, a hydraulic or electricservo drive rotates the core and the part by 180 degrees (or 120 degrees for a three-shot part) allowing alternating polymers to be injected. This is the fastest and most common method because two or more parts can be molded every cycle. Utilities for the rotating mold (i.e., cooling water, compressed air, or special heating) are connected through a central rotary union. Rotational capability can be built into the mold or the machine platen. If there is a family of

Figure 9.40 The rotating mold that is used to produce multi-component injection molded parts in an automatic fashion. (From *Plastic Part Design for Injection Molding—An Introduction*, Malloy, R. A., (1994), Hanser, Munich, p. 396, Fig. 6.82.)
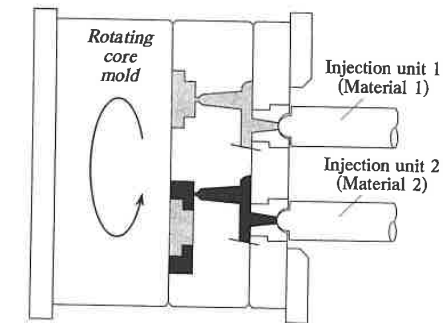
parts that requires rotational transfers, it is more economical to have the rotational capability built into the machine (and pay for it once) than it is to buy a family of molds with the capability in each.

Another variation of multicomponent injection molding involves automatically expanding the original cavity geometry using retractable (movable) cores or slides while the insert is still in the mold. This process is called *core-pull* or *core-back*, as shown in Fig. 9.41. To be specific, the core retracts after the insert has solidified to create open volume to be filled by the second material within the same mold.

9.11.2 Process Advantages

Multicomponent injection molding offers an innovative approach that combines several components inside the mold, thereby, eliminating the need of secondary assembly and reducing the overall manufacturing cost. This process allows a new
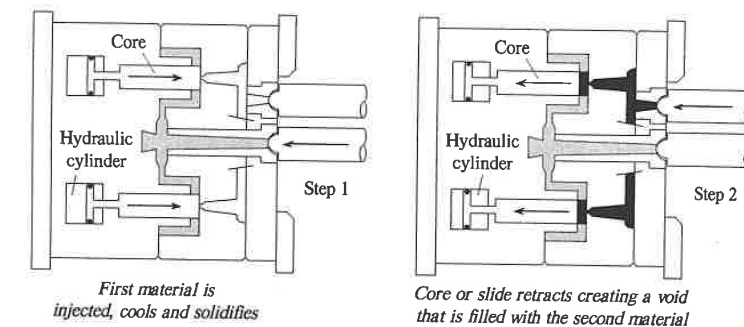

Figure 9.41 Multi-component injection molding using "core pull" or "core-back" technique. (From *Plastic Part Design for Injection Molding—An Introduction*, Malloy, R. A., (1994), Hanser, Munich, p. 396, Fig. 6.83.)

Figure 9.42 (Top) TPE (thermoplastic elastomer) overmolded on ABS/PC for a mobile phone to provide soft touch feeling and drop protection. (Bottom) Black color ABS with transparent acrylic, used on audio push button for light on transmitting. (Parts courtesy of Yomura Company Ltd., Taiwan.)

degree of freedom in industrial design and improves part esthetics, value, quality, and functions (cf. Fig. 9.42). Furthermore, the capability of multicomponent production with a single machine eliminates handling between processes, resulting in savings in part handling, in losses during assembly, and in the amount of floor space required [25].

9.11.3 Process Disadvantages

Process disadvantage is the investment for a more complex (rotating) mold and a special injection molding machine with multiple injection units and a special control system. The processes also require transferring of the insert to different mold or relatively complex tools and processing operations. It should be noted, however, that cost barriers to multicomponent injection molding have fallen as a result of molding equipment being modularized by machine manufacturers [91].

9.11.4 Applicable Materials

Material selection is vital for multicomponent injection molding. A thorough analysis needs to be conducted to determine material compatibility, chemical- and wear-resistance, environmental performance, and other program-specific requirements. Various material combinations will result in widely different levels of adhesion

between the base material and subsequent overmolded materials. It is possible to achieve bonds that range from no-bond to true chemical bonds, where the materials interact at molecular levels and yield superb bond strength, along with chemical and environmental durability. Because multicomponent injection molding involves bonding of different materials into a single consolidated part, adhesion between pairing materials is important. Factors that influence adhesion include compatibility, process temperature, surface contact area and texture, molding sequence, and the design of mechanical interlock systems [25]. One of the most popular applications is the overmolding of a flexible thermoplastics elastomer (TPE) onto a rigid substrate to create the soft-touch feel and improved handling in a finished product. Table 9.3 provides basic guidelines on material adhesion characteristics for a wide range of material combinations; however, it is highly recommended to check with the resin supplier to confirm the material selection and compatibility.

9.11.5 Typical Applications

Complex parts in a variety of applications, from electrical, consumer, industrial, to automotive, are being molded with multicomponent injection molding. Figure 9.43

Figure 9.43 Examples of multi-component injection molded parts by Phillips Plastics Corporation, Hudson, Wisconsin, and Ingersoll Rand, Rockford, Illinois, USA. (Reprinted by permission of the Appliance Manufacturer Magazine.)

Table 9.10   Typical Multi-Component Injection Molding Applications [25]

Automotive	Nonautomotive
Lock housing	Toothbrush
HVAC vent grille	Tool handles
Airbag connectors	Articulating toys
Door rub strip and seal	Appliance knobs
Air duct adjustment wheel	Cosmetic compact
Air duct flap (no adhesion)	Hinge (no-adhesive)
Multi-colored taillight lenses	Electric shaver housing
	TV/VCR remote control unit
	Electric toothbrush cam assembly

shows a number of multi-component injection molded parts and products with typical applications listed in Table 9.10.

9.12   Multiple Live-Feed Injection Molding

The multiple live-feed injection molding process is a technique that applies macroscopic shears at melt–solid interfaces to control the microstructure of materials for producing high-integrity products. The oscillating force exerted at multiple polymer entrances comes from the action of pair(s) of pistons moving in the channels of a special processing head between the barrel and the mold. The melt oscillation keeps the material in the gates molten, whereas layers of different molecular or fiber orientation are being built up in the molding due to solidification. This process provides a means of making parts that are free from voids, cracks, sink marks, and weld-line defects. It also permits tailored orientation of the fibers for fiber-reinforced materials to obtain strengths greatly beyond what conventional injection molding can achieve. Multiple live-feed injection molding is also called *shear controlled orientation injection molding* (SCORIM) [92].

9.12.1   Process Description

The multiple live-feed injection molding requires a special processing head that is mounted onto the end of the injection screw barrel and adapter rings. This head has two double-acting hydraulic cylinders—one for each branch of the split molten stream (cf. Fig. 9.44). The head allows each melt stream to fill a chamber connecting the machine nozzle with the runners and gates. Each chamber has independently controlled pressures that may be alternately applied and removed to each branch of
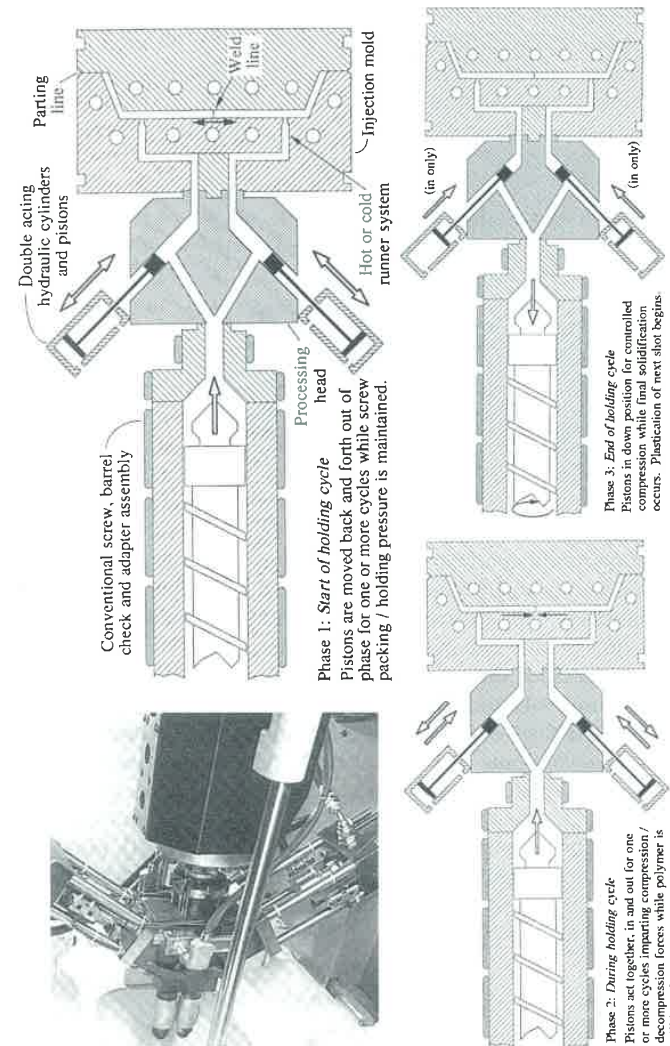
Figure 9.44 Multiple live-feed injection molding provides three different pressure modes that dynamically move and pack the polymer melt inside the mold. (From *Plastic Part Design for Injection Molding—An Introduction*, Malloy, R. A., (1994), Hanser, Munich, p. 60, Fig. 250.)

the split stream. The pistons provide programmed driving forces that dynamically move the molten plastic in the mold. Up to four pistons can be used in each processing head. To fit into existing injection molding machines, this processing head requires approximately 35 cm (14 inches) of space between the barrel of the machine and the mold. It also needs a hydraulic system to activate the hydraulic cylinders and pistons.

During the molding, the mold is first filled as it is in the conventional process. To be specific, the molten polymer is injected from the barrel into the mold through one or both piston channels (cf. Fig. 9.44). The pistons are initially withdrawn from the channel in order to permit the flow of the melt. Once the material starts to cool from the outside and inward, the pistons are activated in the programmed sequence. At first, the piston actions develop the fluctuating melt pressures (with pairing pistons oscillating 180 degrees out of phase) that move and shear the melt in the cavity and in the gates. The pistons then oscillate in phase, providing compression and decompression force to the melt. Meanwhile, new material is introduced to compensate for shrinkage and voids. At the last stage, the pairing pistons compress melt with equal constant pressure, as illustrated in Fig. 9.44.

The programmed sequences of forces drive the "live" melt in the cavity to eliminate or minimize physical imperfections in the part and change the morphology by aligning molecules and reinforced fibers in the preferential direction. This reportedly results in highly efficient packing and enhanced physical properties and performance of the molded product [92,93]. For example, fibers can be aligned circumferentially if a ring is molded, thus greatly enhancing the hoop strength. The desired directions of fiber alignment are decided during component design, and the mold will then be gated accordingly. Figures 9.45 and 9.46 show the micrographs of the resulting fiber orientation using SCORIM process and the comparison of fiber orientation at the weld line using both conventional and the SCORIM processes, respectively.

The multiple live-feed injection molding technology has been extended to two injection units and two double-line-feed devices. By sequencing and timing of the pairing pistons, a multiplayer laminated structure can be formed [94]. Moreover, the multiple live-feed injection molding technology has been employed in conjunction with "Bright Surface Molding (BSM)" for removing surface weld lines from highly reflective aluminum pigmented polypropylene injection molded parts [95].

9.12.2   Process Advantages

Multiple live-feed injection molding introduces programmed sequence of melt movement to create uniform distribution of polymer and/or desirable alignment of reinforcing fibers throughout the mold (cf. Figs. 9.45 and 9.46). Primary benefits of this process, therefore, are greater dimensional stability, part stiffness, and impact strength while eliminating product defects such as voids, cracks, sink marks, and weld-lines. Other benefits include additional design flexibility for thinner walls, fewer ribs and supports, tighter tolerances, and more cost-effective materials. As an example, Table 9.11
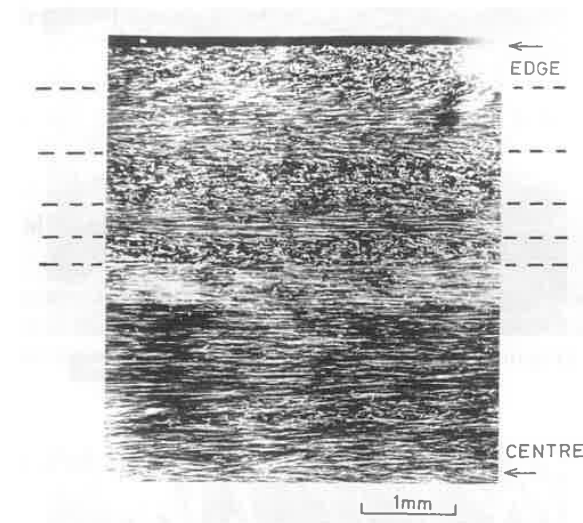
Figure 9.45 The multiple live-feed injection molding process permits controlled orientation of reinforcing fibers in the molded part. This micrograph shows the formation of shear-oriented layers and increased fiber orientation due to changing direction of polymer flow. (Courtesy of Cinpres Limited, Tamworth, England.)

Table 9.11   Comparison of Mechanical Properties of Molded Parts Using Conventional Injection Molding and SCORIM Processes [93]

Material	Weight (% of HA*)	Conventional injection molding			Multiple live-feed injection molding		
		Modulus at 1% strain (GPa)	Tensile strength (MPa)	Strain at break, ε_r (%)	Modulus at 1% strain (GPa)	Tensile strength (MPa)	Strain at break, ε_r (%)
HMWPE	0	1.20 ± 0.08	25.0 ± 0.5	13.0 ± 0.2	4.54 ± 0.61	83.6 ± 5.5	20.8 ± 4.2
HMWPE + HA	50/30	4.03 ± 0.12	39.2 ± 0.7	5.9 ± 0.4	7.45 ± 0.21	73.5 ± 4.1	18.7 ± 3.1
SEVA-C	0	1.81 ± 0.16	35.6 ± 3.2	5.3 ± 0.5	2.97 ± 0.23	41.8 ± 0.8	7.1 ± 2.0
SEVA-C + HA	50/30	5.24 ± 0.34	35.7 ± 5.6	1.8 ± 0.3	7.09 ± 0.21	42.8 ± 3.1	0.5 ± 0.1
S/CA	0	3.24 ± 0.31	70.2 ± 4.8	5.5 ± 1.2	5.81 ± 0.16	98.7 ± 6.8	8.6 ± 2.0
S/CA + HA	30/30	4.94 ± 0.36	60.4 ± 0.4	4.1 ± 0.3	8.63 ± 0.12	65.4 ± 5.3	3.1 ± 0.2

* HA (Hydroxylapatite) percentage for achieving maximum stiffness in conventional and multiple live-feed injection molding, respectively.
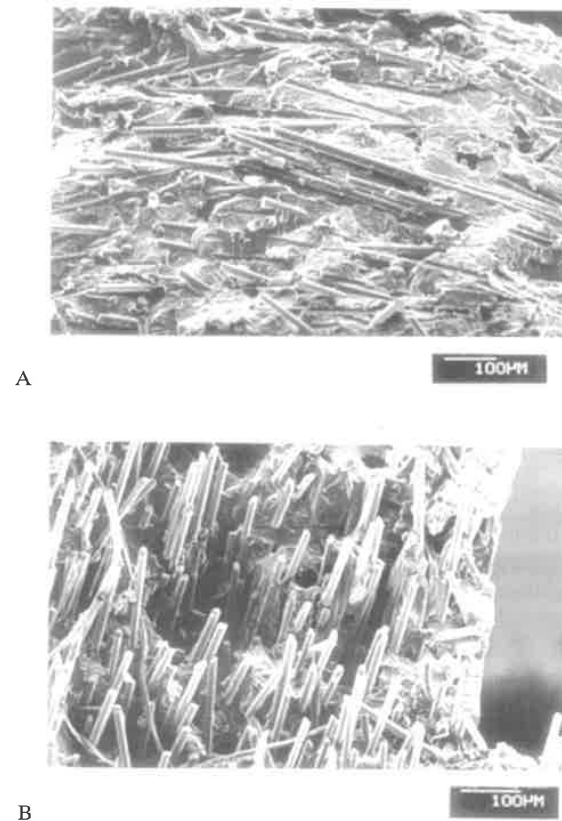
A

B

Figure 9.46 The fracture failure surfaces at the point of a weld line for a conventional sample (top) and a SCORIM sample (bottom). Note the change of fiber orientation at the weld line formation point. (Courtesy of Cinpres Limited, Tamworth, England.)

lists the comparison of mechanical properties of parts using conventional injection molding and SCORIM processes.

9.12.3   Process Disadvantages

As with several other special injection molding processes, the disadvantages of multiple live-feed injection molding is the requirement of the additional processing units, its packing pistons, plus the determination of pairing gates and associated process control parameters for oscillation of melt.

9.12.4   Applicable Materials

The process can be used for a variety of unreinforced and reinforced thermoplastics, thermosets, and liquid crystalline polymers (LCPs), including polyolefins, POM, PA, polyesters, PEEK, polyether sulfone (PES), and polyphenylene oxide (PPO).

9.12.5   Typical Applications

This process has been used to produce orthopedic devices such as prostheses, medical implants like bone fixation and fixation plates [93], and products that permit application of oscillating force at multiple gates to improve part integrity and strength.

9.13   Push–Pull Injection Molding

The push–pull injection molding process is a melt oscillation technique that creates orientation effects in melts containing fibers or LCPs. In this process, the material is pushed in and out of the mold between two injection units during injection and solidification stages. This process results in molecular and fiber orientation along the direction of the movement. It also minimizes effect of weld lines by dispersing them throughout the part and eliminates voids, cracks, and microporosities in large cross-section molding.

9.13.1   Process Description

The push–pull injection molding process is very similar to multiple live-feed injection molding (see Sec. 9.12). It was originally unveiled by Klockner Ferromatik Desma at the K'89 show [96]. As shown in Fig. 9.47, the push–pull injection molding system includes two injection systems and a two-gate mold [90]. The master injection unit injects the material into the cavity through one gate. It overflows the cavity and the extra material is pushed into the secondary injection unit, where the screw recoils 10 to 15 mm to make room for the material [97]. This process is then reversed with material being pulled back to the master injection unit, and the cycle repeats.

As the material flows back and forth through the mold, molecular orientation is continuously created and subsequently locked in as the material solidifies from the outer layers toward the hot core. By keeping the molten polymer in laminar motion during solidification, the molded parts acquire an oriented structure throughout the volume. If the mold is complex and the melt has to flow around obstacles, the motion will create better mixing in the "eddy current" area behind the obstacle and reduce the weakening effect of the weld lines.
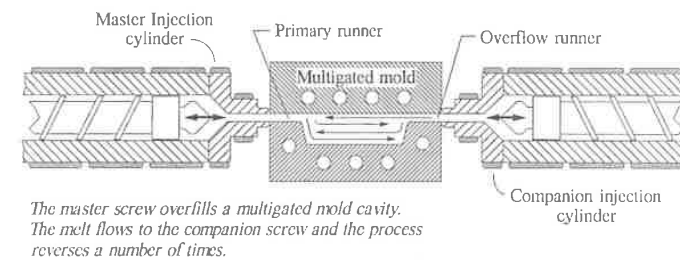
Figure 9.47 Principle of push–pull injection molding. (From *Plastic Part Design for Injection Molding—An Introduction*, Malloy, R. A., (1994), Hanser, Munich, p. 61, Fig. 2.51.)

    Although 10 repetitions of push–pull cycles are standard in the production of the molded part, the push–pull sequence reportedly can be repeated as many as 40 times. The ram (screw) speeds for both injection units can be adjusted to generate different pressures inside the cavity. The injection cycle of push–pull injection molding is apparently, considerably longer than is that of conventional injection molding. Because the material is solidifying as the push–pull cycles take place, however, the filling and packing stages are virtually merged into one single stage.

    A modification of the previously mentioned system with two injection units and four gates has been developed to produce materials with isotropic properties. The flow is basically activated in orthogonal fashion with two pairs of opposite gates opened and closed alternatively. Other developments of this process include a degassing step to prevent material degradation, variation of cycle time and sequence to compensate for cooling gradients, and using nitrogen to create hollowed parts similar to gas-assisted injection molding [96,98]. As examples, Fig. 9.48 shows a car stabilizer component using a hybrid push–pull and gas-assisted injection process. The material is a partial aromatic polyamide (polyarylamide) with 50% short glass fiber content. Six push–pull strokes were applied before gas injection. Figure 9.49 shows the parallel cut of a plastic component made by the hybrid push–pull and gas-assisted injection process.

9.13.2   Process Advantages and Disadvantages

The process advantages and disadvantages of push–pull injection molding are similar to those of the multiple live-feed injection molding (see Sec. 9.12).

9.13.3   Applicable Materials

Glass fiber reinforced PAs are materials well suited for push–pull injection molding. This process increases their stiffness by 30% along the direction of the melt flow.

Figure 9.48 A car stabilizer component using a hybrid push–pull and gas-assisted injection process. The material is a partial aromatic polyamide (polyarylamide) with 50% short glass fiber content. (Photo courtesy of StructoForm GmbH, Aachen, Germany.)

Push–pull injection molding has been applied to LCPs and glass fiber–filled LCPs [98]. Increased orientation of the LCPs has boosted an increase of tensile strength of 150% and toughness increase of 250% over the material supplier's literature [25]. In another application, push–pull injection molding can also eliminate the mechanical anisotropy of LCPs by making them orthotropic [99]. This process has also been applied to several high-performance thermoplastics, such as PES, PPS, PPA, PEK, and PEAK.
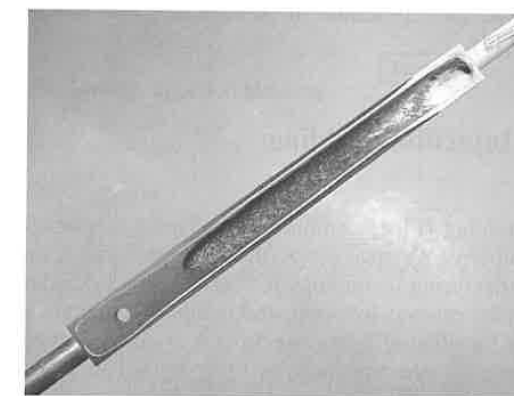

Figure 9.49 A car stabilizer component using a hybrid push–pull and gas-assisted injection process. The material is a partial aromatic polyamide (polyarylamide) with 50% short glass fiber content. (Photo courtesy of StructoForm GmbH, Aachen, Germany.)

Figure 9.50 Airbus window frame made from LCP Vectra A using push–pull injection molding. (Photo courtesy of StructoForm GmbH, Aachen, Germany, with contributions from Klöckner Ferromatik, Germany, TU Delft, The Netherlands, IKV Aachen, Germany, Aerospatiale, France, and Geiger Technik, Germany.)

9.13.4   Typical Applications

Possible applications of push–pull injection molding are automotive parts, as well as aerospace and electronics components. For example, Fig. 9.50 shows an Airbus window frame using push–pull injection molding. The push–pull injection molding process is also said to have potential in powder injection molding of metals and ceramics (see Sec. 9.14).

9.14   Powder Injection Molding

Powder injection molding (PIM) combines the shaping advantage of the injection molding with the superior physical properties of metals and ceramics. It is rapidly emerging as a manufacturing technology for ceramics and powdered metals, where high volume, high performance, low cost, and complex shape are required. In these processes, a custom formulated mixture of metal or ceramic powder and polymer binder is injected into a mold and form into the desired shape (the "green part"), which is akin to conventional injection molding and high-pressure die casting. After the molding process, the binder is removed and the resulting "brown part" undergoes a sintering process to create metallurgical bonds between the powder particles imparting the necessary mechanical and physical properties to the final part.

9.14.1   Process Description

Powdered metal injection molding and ceramic injection molding are special injection molding processes used to produce highly complex metal and ceramic parts that otherwise would require extensive finish machining or assembly operations [100–104]. Figure 9.51 illustrates the basic steps involved in PIM: mixing for forming feedstock, molding, debinding (binder extraction), and sintering (densification). The last two steps can be combined into a single thermal cycle.

Mixing for Forming Feedstock

Feedstock is the mixture of powder and binder in forms of pellets and granules. Its formation involves mixing selected ceramic raw materials or powdered metal alloys and polymeric binders to form a moldable formulation. Powders are small particles that usually have sizes between 0.1 and $20\,\mu m$ with a nearly spherical shape to ensure desirable densification. In principle, any metal that can be produced as a powder can be processed using this method. One exception is aluminum, which forms an oxide layer on the surface, thus preventing sintering. Commonly used particles are stainless steels, steels, tool steels, alumina, iron, silicates, zirconia, and silicon nitride. Binder is usually based on thermoplastics such as wax or polyethylene, but cellulose, gels, silanes, water, and various inorganic substances are also in use. The binder usually consists of two or three components: thermoplastics and additives for lubrication, viscosity control, wetting, and debinding [100].
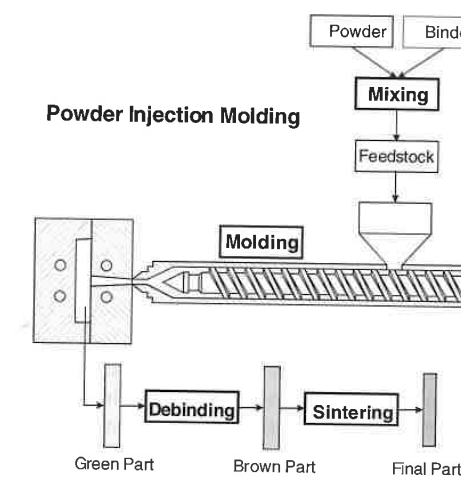

Figure 9.51 Basic steps involved in powder injection molding.

A typical feedstock is composed of approximately 60 vol.% powders and 40 vol.% binders. Twin-screw or high-shear cam action extruders are often used to mix the materials at elevated temperatures (100 to 200°C). Upon exiting the extruder, the mixture is chopped into pellets and granules to be used for injection molding. There are five factors that determine the attributes of the feedstock: powder characteristics, binder composition, powder–binder ratio, mixing method, and pelletization technique. An ideal feedstock is one that is easy to mold and that easily attains the final product dimensions.

Molding

The tooling for powder injection molding is similar to that used in conventional polymer injection molding except that molding is oversized to account for sintering shrinkage. During the molding stage, feedstock pellets are heated to temperatures of 150 to 200°C and injected into the mold at an injection pressure range of 35 to 140 MPa (5000 to 20,000 psi) and with cycle times of 30 to 60 seconds, depending on the part dimension and actual molding formulation employed [101]. The mold can be either heated to control the viscosity of the mixture or be kept at room temperature to facilitate cooling. At the end of the cycle, the molded components (so-called green part) have a strong crayonlike consistency and are suitable for autoejection from the mold via push-out pins or stripper rings. Note that, depending on the binder system used, the molded part could be very brittle; therefore, for critical parts it is desirable to remove the parts from the mold using a handling system.

Debinding (Binder Extraction)

The main objective of debinding is to remove the binder in the shortest time with the least impact on the compact. This step of the process is the most critical step in the powder injection molding process. Debinding is typically accomplished by one of the three general procedures: thermal binder removal method, catalytic binder removal agents, or solvent extraction method [100]. The debinding process, which removes as much as 30 to 98% of the binder, can be a long process. It typically takes many hours or even days, depending on part thickness and powder grain size. After the debinding stage, the resulting compact is called the "brown part."

Sintering (Densification)

The last step in the process is the sintering, which is a thermal treatment for bonding the particles into a solid mass. Furnaces are used at this stage that permit sintering at temperatures close to the material melting temperature under a controlled atmosphere or in a vacuum. Inside the furnace, sintering can be performed in air for ceramics or in a reduced atmosphere for metals to prevent oxidation or to reduce any existing oxides. A simple hydrogen atmosphere is sufficient for many metals. The atmosphere must contain corresponding carbon compounds to achieve

the appropriate carbon content in the steel for steels in which carbon is an essential element.

During the sintering process, intergranular pores within the molded part are removed, resulting in significant shrinkage (typically on the order of 10 to 20%). Meanwhile, the finished part achieves a density of around 97%, at which the mechanical properties differ only slightly, if at all, from those parts manufactured by alternative methods.

9.14.2 Process Advantages

The process advantages of PIM can be summarized in the following:

- Reduced manufacturing cost savings of 20 to 40% over conventional metal and ceramic processing methods [101]
- A net-shape process technology with good dimensional tolerance control and with little or no secondary machining required
- Capability of producing complex shape and geometric features
- High production rates through the use of injection molding and/or automated production
- Parts with mechanical properties nearly equivalent to wrought materials

9.14.3 Process Disadvantages

The major process disadvantages of powder injection molding are:

- Constraints of part size and thickness
- Control of volumetric shrinkage and thickness uniformity

Although there is no technical limit to the maximum size of part that could be produced, two economic considerations restrict the sizes and part thickness. First, the larger the part, the greater the proportion of the overall cost that is attributable to the raw material, which is costly. Second, the thicker the section, the longer the debinding time and, thus, the higher the cost of that part of the process. At present, the limiting thickness is about 30 mm.

The "green part" contains a high-volume percentage of binder (as much as 50%), which leads to significant shrinkage during sintering. It is a major requirement of the sintering process that this shrinkage be controlled. Variations in wall thickness will cause variations in shrinkage during sintering, which makes dimensional control difficult. In addition, uniform wall thickness is critical in order to avoid distortion, internal stresses, voids, cracking, and sink marks.

Figure 9.52 A range of powder injection molding components [104].

9.14.4   Typical Applications

PIM is applicable to a variety of automotive, consumer, electronic, computer peripherals, medical, industrial, military, and aerospace components. Automotive applications include turbocharger, brake, and ignition components, as well as oxygen sensors. Figure 9.52 shows a range of PIM components [104].

9.15   Reaction Injection Molding

Reaction injection molding (RIM) involves mixing of two reacting liquids in a mixing head before injecting the low-viscosity mixture into mold cavities at relatively high injection speeds. The liquids react in the mold to form a cross-linked solid part. The short cycle times, low injection pressures, and clamping forces, coupled with superior part strength and heat and chemical resistance of the molded part make RIM well suited for the rapid production of large, complex parts, such as automotive bumper covers and body panels.

9.15.1   Process Description

Reaction injection molding (RIM) is a process for rapid production of complex parts directly from monomers or oligomers. Unlike thermoplastic injection molding, the

shaping of solid RIM parts is through polymerization (cross-linking or phase separation) in the mold rather than solidification of the polymer melts. RIM is also different from thermoset injection molding in that the polymerization in RIM is activated via chemical mixing rather than thermally activated by the warm mold. During RIM process, the two liquid reactants (e.g., polyol and an isocyanate, which were the precursors for polyurethanes) are metered in the correction proportion into a mixing chamber where the streams impinge at a high velocity and start to polymerize prior to being injected into the mold (cf. Fig. 9.53). Due to the low-viscosity of the reactants, the injection pressures are typically very low even though the injection speed is fairly high. Because of the fast reaction rate, the final parts can be de-molded in typically less than one minute. Table 9.12 provides a brief comparison between RIM and thermoplastic injection molding.

There are a number of RIM variants. For example, in the so-called reinforced reaction injection molding (RRIM) process, fillers, such as short glass fibers or glass flakes, have been used to enhance the stiffness, maintain dimensional stability, and reduce material cost of the part. As another modification of RIM, structural reaction injection molding (SRIM) is used to produce composite parts by impregnating a reinforcing glass fibermat (preform) preplaced inside the mold with the curing resin. On the other hand, resin transfer molding (RTM) is very similar to SRIM in that it also employs reinforcing glass fibermat to produce composite parts; however, the resins used in RTM are formulated to react slower, and the reaction is thermally activated as it is in thermoset injection molding.
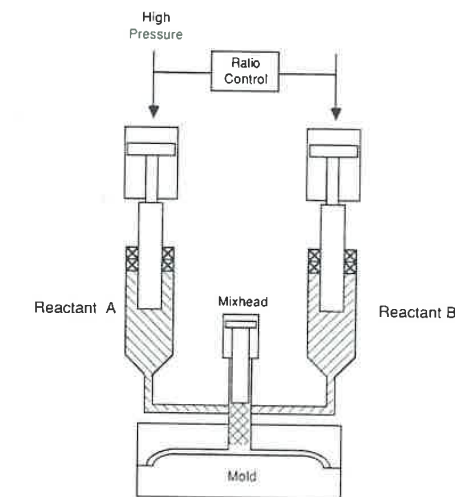

Figure 9.53 Schematic diagram of a reaction injection molding machine. (From *RIM Fundamentals of Reaction Injection Molding—Introduction*, Macosko, C. W., (1988), Hanser, p. 2, Fig. 1.1.)

Table 9.12   Comparison between RIM and Thermoplastic Injection Molding [25]

	RIM	Thermoplastic injection molding
Material processing temperature (°C)	40–60	150–370
Material viscosity (Pa-s)	0.1–1.0	10^2–10^5
Injection pressure (bar)	100–200	800–1000
Clamp force requirement per m^2 (metric ton)	45	1800–2700

9.15.2   Process Advantages

The process advantages of RIM stem from the low pressure, temperature, and clamp force requirements that make RIM suitable for producing large, complex parts with lower-cost molds. In addition, the capital investment on molding equipment for RIM is lower compared with that of injection molding machines. Finally, RIM parts generally possess greater mechanical and heat-resistant properties due to the resulting cross-linking structure.

9.15.3   Process Disadvantages

The mold and process designs for RIM become generally more complex because of the chemical reaction during processing. For example, slow filling may cause premature gelling, which results in short shot, whereas fast filling may induce turbulent flow, creating internal porosity. Improper control of mold-wall temperature and/or inadequate part thickness will either give rise to moldability problem or cause scorching of the materials. Moreover, the low viscosity of the material tends to cause flash that requires trimming. Another disadvantage of RIM is that the reaction with isocyanate requires special environmental precaution due to health issues. Finally, like many other thermosetting materials, the recycling of RIM parts is not as easy as that of thermoplastics.

9.15.4   Applicable Materials

Polyurethane materials (rigid, foamed, or elastomeric) have traditionally been synonymous with RIM as they and ureaurethanes account for more than 95% of RIM production. Alternative RIM materials include nylon (NYRIM), dicyclopentadiene (DCPD-RIM), acrylamate/acrylesterol, epoxies, unsaturated polyesters, phenolics, thermoset resins, and modified polyisocyanurate.

9.15.5   Typical Applications

Due to the low-viscosity of the reacting liquid mixtures, RIM is generally used to produce large, complex parts, especially for exterior and interior automotive components such as bumper beams, bumper fascias, and door panels. Other automotive applications include fenders, exterior trims, arm rests, steering wheels, and window gaskets. Nonautomotive applications range from furniture, business machine housings, medical and industrial enclosures, agricultural and construction parts, to appliances and recreational equipment.

9.16   Resin Transfer Molding and Structural RIM

Resin transfer molding (RTM) and SRIM are two similar liquid composite molding (LCM) processes that are well suited to the manufacture of medium-to-large, complex, lightweight, and high-performance composite components primarily for aerospace and automotive industries. In these processes, a reinforcement fiber mat (preform) is preplaced in a closed mold to be impregnated by a low-viscosity, reactive liquid resin in a transfer or injection process. These two processes differ in such areas as the resins used, mixing and injection set up, mold requirement, cycle time, fiber volume fraction, and suitable production volume.

9.16.1   Process Description

LCM processes such as resin transfer molding (RTM) and structural reaction injection molding (SRIM) are recognized as the most feasible and structurally efficient approach to mass produce lightweight, high-strength, low-cost structural composite components. In general, these processes consist of preparation of a reinforcing fiber mat (known as preform), preplacement of the dry preform in a closed mold, premixing and injection or transfer molding of reactive liquid resin (cf. Fig. 9.54), impregnation of the fiber mat by the curing resin, and removal of the cured, finished component from the mold (demolding).

The preform is the assembly of dry (unimpregnated) reinforcement media that is preshaped (via, e.g., thermoforming) and assembled with urethane-formed cores, if necessary, into a three-dimensional skeleton of the actual part. Both of the RTM and SRIM processes make use of a wide variety of reinforcing media, such as woven and nonwoven fiber products, die-cut continuous strand mats (CSM), random mats consisting of continuous or chopped fibers laid randomly by a binder adhesive, knit braiding or two- and three-dimensionally braided products, or hybrid preform made of layers of different types of media. The selection of the preform architecture depends on the desired structural performance, processability, long-term durability, and cost.
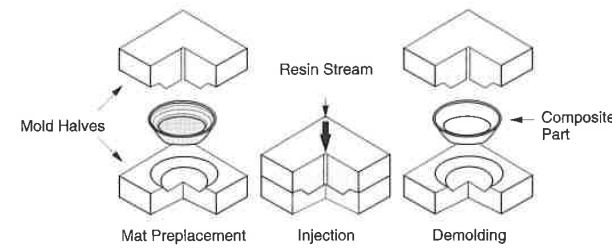
Figure 9.54 Schematic of the resin transfer molding (RTM) and structural reaction injection molding (SRIM) processes [105].

The main difference between RTM and SRIM stems from the reaction activation mechanism of the resins used [106]. This leads to different filling and cycle times, mixing and injection set-up, volume fraction and construction of the reinforcement, mold requirements, and suitable production volume. To be more specific, the chemical systems used in reaction molding processes can roughly be divided into two types: thermally activated and mixing activated. The resins used in RTM fall into the thermally activated category, whereas the resins used in SRIM are mixing activated. In the RTM process, a static mixer is used to produce the mixture at a typical mix ratios of 100 to 1 by volume. These thermally activated resin systems do not react appreciably at the initial resin storage temperature. They reply on heated mold wall to accelerate the chemical reaction. As a result, the filling times for RTM can be as long as 15 minutes, and the cycle time is on the order of an hour or longer, depending on the resin and application. Given the slower injection and reaction rates with RTM, high-volume fraction and more complex preform can be used to improve the part strength and performance further. In addition, the low viscosity of the resin and the slow injection rate result in low injection-pressure and clamp-force requirements. As a result, RTM allows the use of so-called soft tool (e.g., wood-backing epoxy mold or aluminum mold). Because of the long cycle time, RTM is generally limited to a low-volume production (i.e., less than 10,000 parts).

On the other hand, SRIM derives its name from the RIM process (cf. Sec. 9.15), from which its resin chemistry and injection techniques are adapted. That is, the chemical reaction is activated by impingement mixing of two highly reactive components in a special mixing head under high pressure. Upon mixing, the mixture is subsequently injected into the mold at a lower pressure. The resin starts to cure as it impregnates the preform and forms the matrix of the composite. Due to the fast reaction rate and rapid build up of viscosity from curing, the cavity has to be filled within a few seconds. In addition, the cycle time is as short as 1 minute. Flow distances for typical SRIM applications are therefore limited to 0.6 to 0.9 m from the inlet gate. Furthermore, the volume fraction and construction of the reinforcement have to be selected carefully to facilitate fast, complete filling before gelation occurs. Because of the high injection rate and short cycle time, SRIM generally uses steel tool and is suitable for medium- to high-volume production (10,000 to 100,000 parts).

9.16.2 Process Advantages

Prepreg/autoclave process has traditionally been the major manufacturing technique for producing lightweight composite components in the aerospace industry. This process however, is slow, expensive, and labor intensive. In addition, common manufacturing processes in automotive industry, such as thermoplastic injection molding and compression molding of SMC, can only incorporate low-volume, short fiber reinforcement. Thus, they cannot produce high-strength products needed for structural applications. RTM and SRIM, therefore, offer viable options for producing lightweight, high-performance structural components for both aerospace and automotive industries.

In addition, low injection pressure is another process advantage of both RTM and SRIM. Although the injection pressure varies with the permeability of the fiber mat, part geometry, and the injection rate, typical injection pressure varies from 70 to 140 kPa for low injection rate and reinforcement content (10 to 20%) to 700 to 1400 kPa for rapid mold filling and high reinforcement content (30 to 50%) [105]. Furthermore, the RTM and SRIM processes employ closed mold, which reduces or eliminates the emission of hazardous vapor. Other advantages include more repeatable part thickness and minimal trimming and de-flashing of the final part.

9.16.3 Process Disadvantages

The RTM and SRIM processes are more suitable for producing large, structural components with limited areas of intricate details. Tiny details, such as grooves or slots for assembly, as well as ribbed structures or bosses are not easy to mold in due to the difficulty involved in preform preparation and preplacement of preform for these features [105]. In addition, it is challenging to provide a Class A surface with standard resins and structural reinforcement materials, unless gel coats are used.

9.16.4 Applicable Materials

Of the several resin systems employed by RTM, polyesters are the mostly used resins because of the low cost. Other resins used in RTM include epoxies, vinyl esters, acrylic/polyester hybrid, acrylamate resin family, and methymethacrylate vinyl esters. On the other hand, common resins used for SRIM include urethane, acrylamate, and dicyclopentadiene.

9.16.5 Typical Applications

Typical applications for RTM and SRIM include automotive components (grille opening panels, hoods, deck lids, doors, and structural cross members), sport equip-

Table 9.13   Common RTM Applications [25]

Automotive	Others
Doors	Building doors
Panels	Cash dispensers
Roof panels	Chemical pumps
Spoilers	Electrical covers
Truck cabs	Instrument cases
Wind deflectors	Lifeboat covers
Aerospace	Machine covers
Aircraft engine blades	Manhole covers
Aircraft interior panels	Marine hatches
Aircraft wing bases	Medical equipment
Sport Equipment	Plumbing components
Bicycle frames	Railway body panels
Bicycle handlebars	Railway seats
Boat hulls	Roof tiles
Canoe paddles	Sinks
	Storage tanks
	Traffic light boxes

Table 9.14   Common SRIM Applications [25]

Automotive and Truck	Other
Bumper beams	Copier heat shields
Door panels	Office chair shells
Instrument panels	Satellite antenna disks
Landau roofs	
Noise shields	
Rear window decks	
Spare tire covers	

ment, marine, medical, construction, and corrosion resistance (for chemical industry). Table 9.13 lists the various RTM applications and Table 9.14 those for SRIM [25].

9.17   Rheomolding

Rheomolding is one of the melt-flow oscillation techniques that employs melt vibration (typically at low frequencies) to reduce the viscosity during filling. It also helps to alter the molecular orientation, thereby improving the mechanical properties, stiffness, strength, and clarity of the molded parts without resorting to processing aids such as thinning and nucleating agents.

(Note: in the area of die-casting process with metals, Rheomolding is sometimes referred to as a special process that employs semi-solid alloys as the material charge. The semi-solid alloys are generated by mechanically agitating solidifying liquid metals or by means of vibration.)

9.17.1   Process Description

Melt-flow oscillation or melt vibration can be applied in various forms to modify the mechanical, rheological, and/or optical properties of the molded materials [107]. For instance, melt vibration using mechanical shaking/vibration or ultrasonic vibration devices can homogenize, thus increasing the density of the molded material. In addition, vibration reduces the melt viscosity and changes the relaxation kinetics, thereby influencing diffusion and rate-sensitive processes such as nucleation and growth of crystals, blending, and orientation. Furthermore, vibration can also generate heat locally by internal friction, resulting in fusion at weld lines, reduction of surface stresses, orientation birefringence, and frictional coefficient at the wall as well as increase in the throughput. Other melt-flow oscillation or melt-vibration technologies, such as multiple live feed injection molding, push–pull injection molding, and vibration gas injection molding, are discussed in Secs. 9.12, 9.13, and 9.20, respectively. Practical applications of the principles of melt vibration have been implemented in injection molding, extrusion, compression molding, and thermoforming processes.

The underlying principle of Rheomolding is based on the fact that the material rheology is a function of vibration frequency and amplitude in addition to temperature and pressure [108,109], a phenomenon manifested itself as the more well-known shear-thinning behavior of polymer melts. It has been reported that low-frequency and high-amplitude oscillations, coupled with traditional cooling treatments, affect plastic materials in much the same way as does faster cooling [108]. This control of material's cooling rate can be used to manipulate the material orientation (for amorphous materials) and morphology (or semi-crystalline materials) for better material properties.

Rheomolding is produced by the reciprocating action of one or more melt accumulator pistons adjacent to the melt-flow path. As illustrated in Fig. 9.55, an add-on actuator is inserted between the barrel of the injection machine and the mold to provide the melt vibration. With such a configuration, the melt plastication and the injection vibration occur separately. In an alternative design, the mold is modified to incorporate small piston(s) to apply vibration directly to where it is needed, such as weld lines or other critical areas (cf. Fig. 9.56). In the Rheomolding processes, both hydrostatic pressure and shear force are oscillated during cycle time to modify the rheological properties, orientation, and crystallization kinetics of the polymer melts. The frequency, amplitude, and phase shift of the vibration modes are designed based on the material used and the objectives to impart quality and enhance performance.
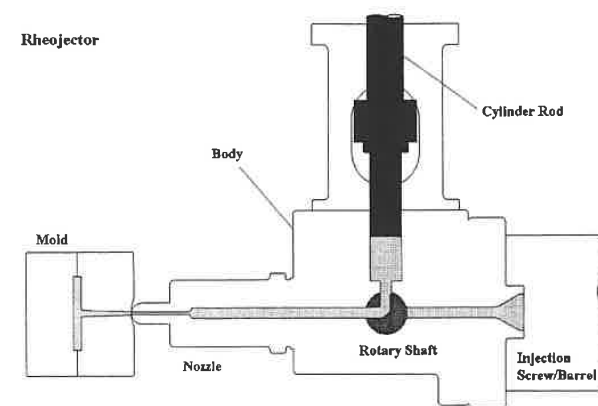
Figure 9.55 An add-on actuator inserted between the injection barrel and the mold applies the melt vibration to the molded material [107]. (Reprinted by permission of the Society of Plastics Engineers.)
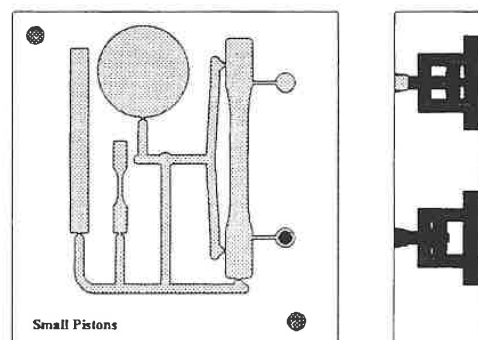

Figure 9.56 An ASTM mold equipped with a set of two pistons to vibrate the melt during filling and packing [107]. (Reprinted by permission of the Society of Plastics Engineers.)

9.17.2 Process Advantages

As with the objectives of other melt-flow oscillation techniques, Rheomolding offers the ability to modify the rheological and orientational states of polymer melts during processing, thus influencing the resulting mechanical (such as tensile strength, modulus, and impact) and optical properties (such as clarity) of the molded parts. It

has been used to eliminate or strengthen weld lines and enhance the properties (by as much as 65 to 120%) [108]. Parts with thinner walls (which uses less material) can therefore be made or less expensive material can be used if the Rheomolding process enhances a specific property. Other process benefits include improved weld-line strength, reduced internal stresses and part warpage, potentially controllable crystallization kinetics, better surface finish, and lower friction coefficient at the wall surfaces.

9.17.3 Process Disadvantages

Similar to many other emerging processes, the know-how of the process has yet to be established to help fully utilize the potential benefits mentioned earlier. In addition, the required hardware and licensing fee adds additional costs to the production. Furthermore, it needs longer development time to design or build in the gas-vibration system and to identify the optimal processing conditions. In addition, there is a limitation to the melt temperature as warm material tends to relax and loses the orientation.

9.18   Structural Foam Injection Molding

Structural foam injection molding is an extension of injection molding for producing parts with cellular (or foam) core sandwiched by solid external skins. This process is suitable for large, thick parts that have enough strength-to-weight ratio to be used in load- or bending-bearing in their end-use applications. Despite the surface swirling pattern, which can be eliminated with special molding techniques or the postmolding finishing operations, this process permits molding of large parts at low pressure and with no sink marks and warpage problems. Structural foam parts can be produced with physical blowing agents like nitrogen gas or chemical blowing agents.

9.18.1 Process Description

Structural foam injection molding produces parts consisting of a solid external skin surrounding an inner cellular (or foam) core (see Fig. 9.57). There are several variants of this process. The most common one is the low-pressure structural foam injection molding. Other additional processes are gas counter pressure, high pressure, co-injection, and expanding mold techniques. As shown in Fig. 9.58, the low-pressure structural foam molding process involves a short-shot injection accompanied by expansion of melt during molding. The melt expansion is created from gas (typically nitrogen) dissolved in the polymer melt prior to injection. It can also be assisted by gas released
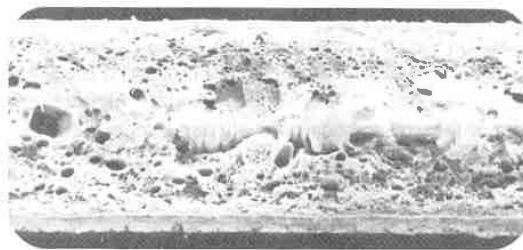
Figure 9.57 Cross-section of a typical structural foam molded part showing the integral skin layers and the foam core. (From *Plastic Part Design for Injection Molding—An Introduction* (1994), Malloy, R. A., Hanser, Munich, p. 115, Fig. 2.109.)
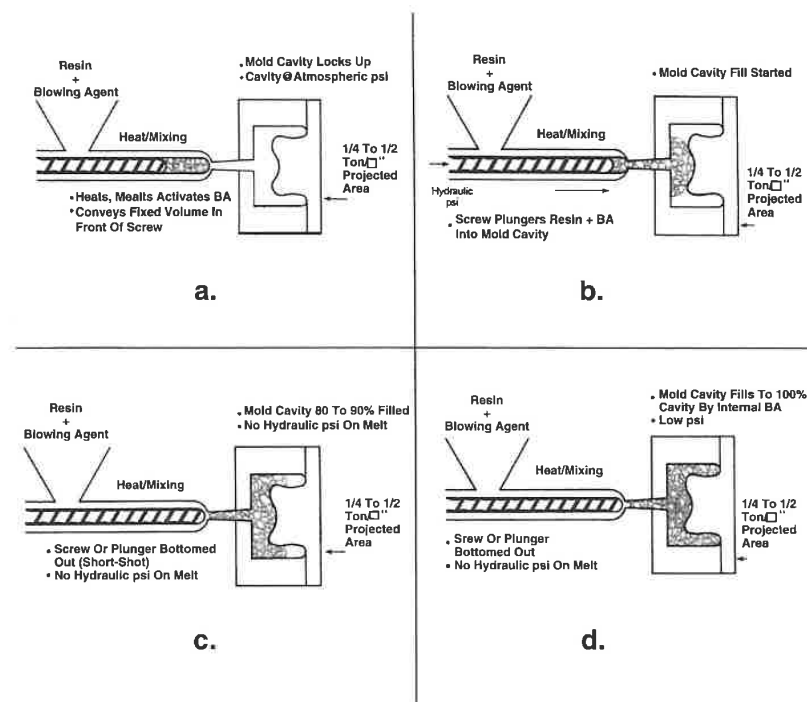

Figure 9.58 Various stages for the structural foam injection molding process. (From *Injection Molding Alternatives—A Guide for Designers and Product Engineers* (1998), Malloy, R. A., Hanser, Munich, p. 63, Fig. 5.5.2.)

from the chemical blowing agents in a resin-compatible carrier blended with the polymer pellets. The melt expansion from the foaming process generates an internal gas pressure of 21 to 34 Bars (2.1 to 3.4 MPa or 300 to 500 psi), which is sufficient to drive the polymer melt continuously to fill the extremities of the cavity. Note that this process requires only 10% of the pressure normally required to fill and pack out the mold with the conventional injection molding process. In addition, structural foam injection molding realizes part weight reduction by replacing plastics with gas and eliminates the typical shrinkage associated with thick parts with the melt expansion. The foam structure provides insulation effect so that the part cools down at a slower rate. Due to the low injection pressure and slow cooling rate (which allows material relaxation), the process produces parts with very low molded-in residual stresses.

As indicated earlier, both nitrogen and chemical blowing agents can be added to the polymer melt to create foam structure. To add nitrogen into the polymer melt and to hold the polymer-gas mixture under pressure until it is injected into the mold, it requires special structural-foam molding equipment with an accumulator. When the external pressure of the melt is relieved as the material enters the mold cavity, the gas begins to emerge from the solution. At the end of a short-shot injection, the pressure decreases rapidly as the material relaxes. Thus, gas expands immediately at this point, pushing the material to fill the rest of the mold. On the other hand, the chemical blowing agents generate gas (nitrogen, hydrogen, or carbon dioxide) by either thermal decomposition or chemical reaction in the melt. Selection of chemical blowing agents depends on the polymer material used, cost, convenience, safety, cell structure, the nature and quantity of gas generated, and the exothermic (heat releasing) or endothermic (heat absorbing) nature of the reaction. If a blowing agent is exothermic, it increases the material temperature and the gas pressure. On the other hand, the endothermic blowing agent helps to cool down the cell structure and reduce internal gas pressure, thereby, allowing the part to be removed in a shorter time. After foaming, the gas inside the core of the part must diffuse to the atmosphere to release the internal pressure [110].

A gas counterpressure technique has been developed and employed to produce parts with smooth, thicker, and nonporous skin surfaces, and more uniform cell density in the core. In this process, the mold is pressurized with an inert gas to create a pressure level of 14 to 34 Bars (1.4 to 3.4 MPa or 200 to 500 psi) during filling. The pressure in the mold must be high enough to prevent the melt from foaming. To maintain the pressure level, the mold must be sealed with O-ring. It may also be necessary to seal ejector pins, sprue bushing, and other moving slides or cores. The control of a sufficient short-shot weight and a critical timing for the counterpressure to be released are also critical. After the injection, the gas is vented and mold is depressurized to allow the expansion of polymer. Because of the counterpressure, the flow length of the material may be reduced by 10 to 20%, and the density reduction is limited to 8 to 10%, as compared with a level of 10 to 15% with the low-pressure structural foam injection molding [25]. Gas counter pressure using a hot runner mold has been reported in Ref. [111] to mold a polystyrene part with surface smoothness substantially better than that of the parts molded under the same processing conditions using low-pressure structural injection molding.

Similar to conventional injection molding, the part performance strongly depends

Table 9.15   Differences between Design Criteria for Conventional Injection Molding and Structural Foam Injection Molding

Criterion	Conventional injection molding	Structural foam injection molding
Wall thickness	Maximum 5 mm No abrupt changes in thickness	Up to 20 mm possible Optimum 4 to 8 mm Abrupt changes in thickness possible, therefore, greater design freedom
Rib thickness	Two thirds of the wall thickness	Any thickness
Surface feature	Smooth or depending on mold	Irregular and structured
Heavy section	Must be avoided because of sink marks	Possible. But longer cooling time is required to prevent "postblow" (local swelling at thick sections due to material expansion after ejection)

on the material, processing conditions, and the part and mold designs. The size and distribution of foam cells depend on the material, the molding technique, and the processing conditions. They have direct influence on the physical properties of the part. Because of the pressure and temperature fields, a variation of cell size exists within the foamed part with large voids concentrating at the center of the part and areas away from the gate, and gradually decreasing cell size toward the solid part surface. In addition, the surface roughness was found to increase progressively in the direction of melt flow from the cold sprue toward the end of the part [110]. The processing conditions found to enhance the surface smoothness of the part were a higher melt temperature, higher mold temperatures, higher short-shot weight, higher injection speed, and lower blowing agent concentration.

Due to the unique volume expansion feature of structural foam injection molding, the design guidelines for this process are different from those of conventional injection molding. Table 9.15 lists the differences in design criteria [112].

9.18.2   Process Advantages

Structural foam injection molding was introduced on a commercial basis in early 1960s [110]. The advantages of this process are:

- Large, complex parts can be produced with low clamp tonnage (one order of magnitude lower) compared with conventional injection molding using solid thermoplastics.
- The process also produces parts with very low mold-in residual stresses, resulting in more dimensionally stable and flatter parts.
- Even though the increased part thickness enhances part rigidity and load-bearing capability, weight reduction as much as 10 or 15% can be realized.

9.18.3   Process Disadvantages

The major drawback of structural foam injection molding is the swirling pattern on part surface. The swirl results when the gas from the blowing agent breaks through the melt front and is trapped between the mold surface and the polymer. To improve surface appearance and quality, secondary operations, such as sanding, priming, and painting, are required. These postmolding operations result in higher part cost. It should be noted, however, that parts with substantially improved surface quality have successfully been produced using gas counterpressure structural foam injection molding and co-injection structural foam, as well as expanding, high-pressure methods [110].

Because of the reduction of thermal conductivity due to the foam structure, the cooling time of structural foam parts is longer than that of the solid plastic part. If the part is removed from the mold before the material is sufficiently cooled, the internal gas pressure will cause blister on the part surface, especially at the thick sections. Moreover, if the part is painted before the internal gas pressure reaches the ambient, atmospheric pressure, blisters will also occur.

It should also be noted that the selection of blowing agents should avoid the possible material degradation from the by-products of the blowing agents during their decomposition. Finally, due to the characteristic cellular structure, the mechanical properties of structural foamed parts can be significantly lower than the solid parts, even though thicker foamed parts are more rigid than are their solid counterpart on a weight-to-weight basis.

9.18.4   Applicable Materials

Table 9.16 lists the variety of materials used for structural foam injection molding. Among these materials, the most common are low-density and high-density polyethylenes for their low cost, ease of processing, chemical resistance, and low-temperature impact strength. Polypropylene is an attractive material for its increased stiffness and chemical resistance, and its reinforced grades exhibit increased temperature resistance. High-impact polystyrene and modified polyphenylene ether (m-PPE), modified polyphenylene oxide (m-PPO), and polycarbonate are commonly found in high-performance applications. The versatility inherent in polyurethane

Table 9.16   Materials Used in Structural Foam Injection Molding [25]

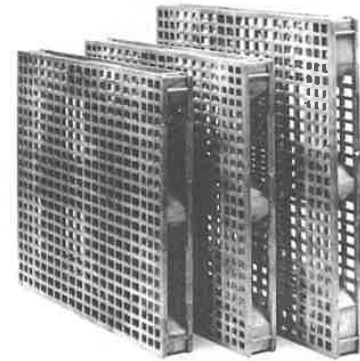|	
Acrylonitrile-butadiene-styrene (ABS)	Polycarbonate
Polyamide 66 (Nylon 66)	Polypropylene
Polyetherimide	Polystyrene
Polyethylene (Low- and high-density)	High-impact polystyrene
Modified polyphenylene ether (m-PPE)	Polybutylene terephthalate
Modified polyphenylene oxide (m-PPO)	Polyurethan

Figure 9.59 Pallets that provide all standard sizes with 13,600 kg (30,000 lb) static load capacity, four-way entry for forklift and two-way entry for floor jacks. Material used is high-density polyethylene [114]. (Reprinted by the permission of the Society of Plastics Industry.)
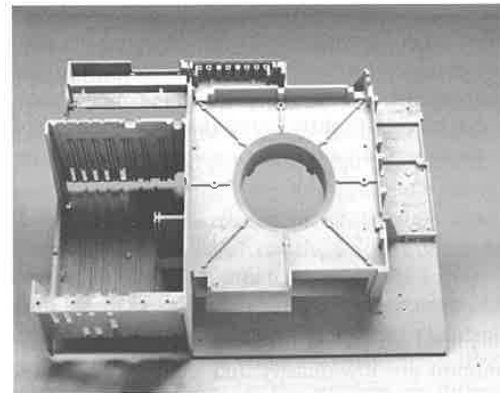

Figure 9.60 Gas chromatograph main frame that supports a 18 kg (40 lb) transformer and withstands temperatures up to 71°C (160°F) from power supply. Material used is modified polyphenylene oxide [114]. (Reprinted by the permission of the Society of Plastics Industry.)
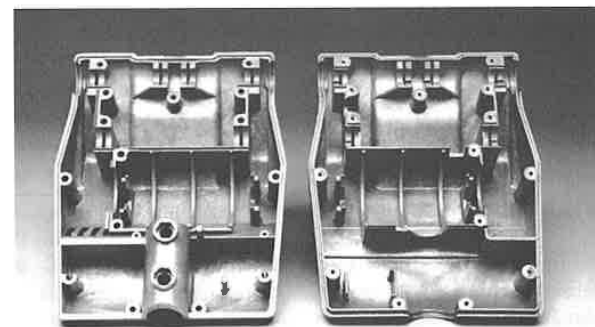
Figure 9.61 Weeder/cultivator that supports motor and gear train and provides chemical resistance to garden chemicals. Material used is 5% glass-filled polycarbonate [114]. (Reprinted by the permission of the Society of Plastics Industry.)
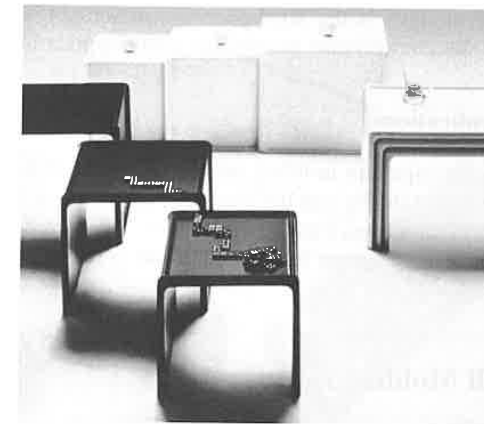
Figure 9.62 Office and home furniture that have structural integrity plus ability to hold tolerances with durable, abrasion-resistant, and colorable finish. Material used is RIM molded rigid polyurethane [114]. (Reprinted by the permission of the Society of Plastics Industry.)
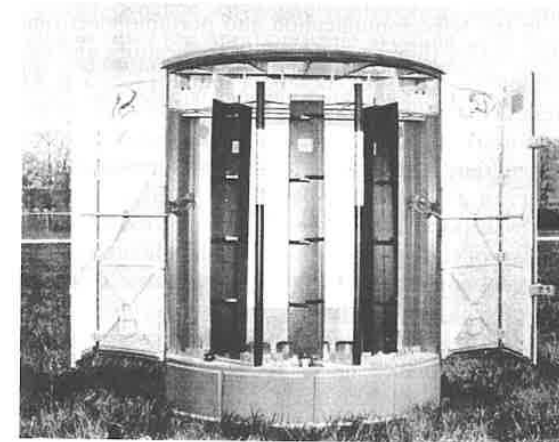
Figure 9.63 A telecommunications housing produced with structural foam injection molding. The material used is foamed polycarbonate. (From Injection Molding Alternatives—A Guide for Designers and Product Engineers (1998), Avery, J. A., Hanser Publishers, Munich, p. 77, Fig. 5.2.16.)

chemistry and the advantages of the RIM process (cf. Sec. 9.15) makes the utilization of polyurethane structural foams possible in many applications [113].

9.18.5 Typical Applications

Because structural foam injection molding enables molding of large, complex parts with rigidity, dimensional stability, and load-bearing capability, it finds many suitable applications in material handling, business machines, automotive components, and medical analysis equipment. Figures 9.59 to 9.63 show a number of structural foam molding applications [114].

9.19   Thin-Wall Molding

Thin-wall (injection) molding is a high-speed, high-pressure injection molding process for producing parts with a nominal wall thickness less than 1.2 mm or flow-length-to-wall-thickness ratios ranging from 100:1 to 150:1 or more. Given the thickness restriction and extreme processing conditions, thin-wall molding has a smaller processing window. Nevertheless, this process becomes increasingly important due to the economic advantages of using thin walls and the unprecedented growth of portable electronic and telecommunication devices that require thinner, smaller, and lighter housings.

9.19.1 Process Description

The rapid growth in the telecommunication and portable electronics markets has created a strong demand for thinner wall-section applications and technology. These hand-held and portable devices, such as cellular phones (see Fig. 9.64), PDAs, and notebook computers, demand light-weighted plastic shells that are much smaller and thinner (less than 1.2 mm), yet still provide the same mechanical strength as conventional parts. Other emerging applications for thin-wall molding are in the manufacturing of medical, optical, and electronic parts or parts that have microscale features. Because decreasing wall thickness realizes part-weight reduction, material savings, and dramatic reduction of cooling time, the thin-wall molding concept is also gaining popularity with cost-conscious end users for large applications like computer monitor housings and automotive instrument panels and fascias.

The term *thin-wall* is relative. Conventional plastic parts are typically 2- to 4-mm thick. Thin-wall designs are called *advanced* when thicknesses range from 1.2 to 2 mm, and *leading-edge* when the dimension is less than 1.2 mm. Another definition of thin-wall molding is based on the flow-length-to-wall-thickness ratios. Typical ratios for these thin-wall applications range from 100:1 to 150:1 or more. Regardless of the definitions, thin-wall parts have a more restrictive flow path compared with conventional injection molded parts [cf. Fig. 9.65(a)]. As a result, they freeze off quickly during molding. To combat this situation, molders often attempt to increase the melt temperature by as much as 38 to 65°C beyond recommended range [115]. Another alternative to avoid premature freeze-off is to inject the material at an order-

Figure 9.64 Plastic housings and components of cellular phones (Parts courtesy of Flambeau Micro, Sun Prairie, Wisconsin, USA). The cellular phone on the top was overmolded with thermoplastic elastomer to provide soft touch feeling and drop protection. (Part courtesy of Yomura Company Ltd., Taiwan.)

of-magnitude–higher injection speed (500 to 1400 mm/s ram speed). It therefore requires very high injection pressures (2400 to 3000 Bars) to achieve fast injection and adequate packing. Given the thickness restriction and extreme processing conditions, thin-wall molding has a smaller processing window as shown in Fig. 9.65(b). Other commonly used techniques to mold thin-wall parts include use of hot runners, multiple gates, and/or dynamic valve-gate control (e.g., sequentially opening and closing valve gates to relay the injection while avoiding weld lines). An optimized ram-speed profile and optimal fill time also help to reduce the pressure requirement [116]. In comparison to a standard injection molding fill time of around 2 s, thin-wall
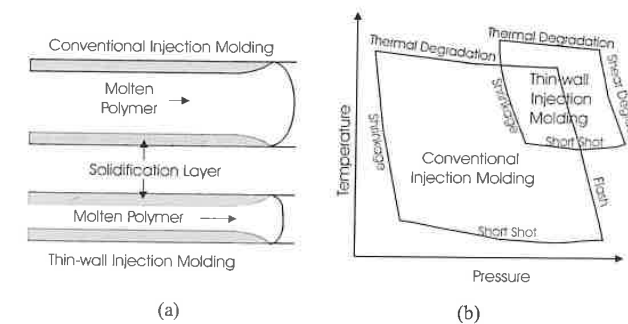

Figure 9.65 Comparisons of (a) flow cross sections and (b) processing window between conventional injection molding and thin-wall injection molding.

molding fill time ranges from 0.1 to 0.5 s. Table 9.17 compares the thin-wall molding with the conventional injection molding applications [117].

Because of the smaller volume required from the thin walls, machines with standard barrel dimension will have a capacity that is too large for the shot volume. As a result, barrels with proper size should be employed to avoid prolonged residence time and excessive melt temperatures that cause thermal degradation. Due to the high velocity and shear rate in thin-wall molding, orientation occurs more readily. To help minimize anisotropic shrinkage in thin-wall parts, it is important to pack the part adequately while the core is still molten. Large gates, greater than the wall thickness, are generally used to ensure sufficient material flow during packing. In addition, gates should be placed in such a manner that the flow occurs from thicker to thinner sections of the cavity. Excessive residence time, melt temperatures, or shear, all of which can cause material degradation, should be avoided.

The ejection system design is more critical for thin-wall molding. First, thin-walled parts are more susceptible to permanent deformation during ejection. Second, to increase the part rigidity, thin-walled parts require reinforcing ribs, which requires higher ejection forces and complicates the ejection system design. Third, the part tends to adhere to the mold wall due to high injection and packing pressures. Finally, if underdesigned, the ejector pins may also buckle or even break during operation, interrupting production and incurring high maintenance costs.

Thin-walled parts for electronic or telecommunication devices should be designed with styling lines and curved surfaces to boost stiffness and enhance part aesthetics. Impact strategies involve using unreinforced plastic housings to absorb the load, using filled thermoplastics to transfer it, or using overmolding process to incorporate a protective thermoplastic elastomer layer (cf. Fig. 9.64). For all cases, the internal components should be fastened snugly, and the design should avoid stress concentration and sharp notches.

9.19.2 Process Advantages

Thin-wall molding realizes part-weight reduction, material saving, and dramatic reduction of cooling time. They also fulfill customers' demand, such as design requirements, cost reduction, and disposal concerns. In particular, conventional injection molding cycle time typically ranges between 30 and 60 s per cycle. Thin-wall molding, however, can run at a cycle time of 6 to 20 s, which cuts the cycle time by two thirds. This reduction in cycle time results in significant manufacturing cost reductions.

9.19.3 Process Disadvantages

Thin-wall molding is more technically challenging than conventional injection molding due to restrictive flow path, excessive processing conditions, and a smaller

Table 9.17 Comparison of Thin-Wall Molding with Conventional Injection Molding [117]

Key factors	Conventional applications	Thin-wall applications
Wall thickness	2.0–3.0 mm (0.08–0.12 in)	1.2–2.0 mm (0.05–0.08 in)	<1.2 mm (<0.05 in)
Machinery	Standard	High-end	Customized high-end equipment
Injection pressure	620–970 Bars (9000–14,000 psi)	1100–1380 Bars (16,000–20,000 psi)	1380–2410 Bars (20,000–35,000 psi)
Hydraulic system	Standard	Standard	Accumulators on injection and clamp units, servo valves
Control system	Standard	Closed loop, microprocessor control over: injection speed, hold pressure, decompression speed, screw RPM, back pressure, all temperatures (including feed throat and oil temperature)	Microprocessor controlled with the following resolutions: speed—1 mm/s, pressure—1 Bar, position—0.1 mm, time—0.01 s, rotation—1 RPM, clamp force—0.1 ton, temperature—1°C
Screw design	Compression ratio: 2.0:1 to 2.5:1, L/D = 20:1 to 24:1, flights 5/10/5: Nitriding not suggested	Compression ratio: 2.0:1 to 2.5:1, L/D = 20:1 to 24:1, flights 5/10/5: Nitriding not typically used	Compression ratio: 2.0:1 to 2.5:1, L/D = 20:1 to 24:1, flights 5/10/5: Nitriding not typically used
Fill time	>2 s	1–2 s	0.1–1 s
Cycle time	40–60 s	20–40 s	6–20 s
Drying	Dew point of –29°C to –40°C; hoppers sizes for material throughput	Dew point of –29°C to –40°C; hoppers sizes for material throughput	Dew point of –29°C to –40°C; hoppers sizes for material throughput
Tooling	Standard	Better venting, heavier construction, more ejector pins, better polish	Extreme venting, very heavy molds, mold interlocks, precise mold surface preparation, extensive ejection features, mold costs 30–40% higher vs. standard

processing window. It also requires rugged, costly tools, and perhaps an upgraded or customized molding machine for high-speed, high-pressure injection. Finally, the high shear rate level that results from the high-speed injection causes shear-induced degradation, as observed by the reduction of failure strain in tensile test results [118].

9.19.4   Applicable Materials

Most of the thermoplastics can be used for thin-wall applications; However, engineering resins, such as PC, acrylonitrile-butadiene-styrene (ABS), PC/ABS blends, and polyamide 6, are more commonly used for thin-wall molding than are commodity resins. This is probably because, as the wall thickness reduces, better physical properties of engineering resins are needed to maintain certain part strength.

9.19.5   Typical Applications

Thin-wall molding is more popular in portable communication and computing equipment, which demand plastic shells that are much thinner yet still provide the same mechanical strength as do conventional parts. Typical applications include cellular phones, pagers, notebook computer housings, medical devices, car stereo faces, minimally invasive surgical devices, electronic connectors, interior and exterior panels for automobiles, and optical storage media with pits on the thin plastic substrate (0.6 mm for DVD).

9.20   Vibration Gas Injection Molding

Vibration gas injection molding (Vibrogaim or vibrated gas-assist molding) is one of the melt-flow oscillation techniques that uses pressurized and vibrated gas to manipulate the polymer melt during processing to modify its flow behavior and to improve the mechanical and optical properties of the molded parts.

9.20.1   Process Description

Vibration gas injection molding is among the latest melt-flow oscillation techniques that provide melt manipulation capabilities during molding [119,120]. Other melt-vibrating technologies, such as multiple live feed injection molding, push–pull injection molding, and rheomolding, have been discussed in Secs. 9.12, 9.13, and 9.17, respectively. Vibration gas injection molding uses the pressurized gas (typically

nitrogen) as the tool or as a "gas spring" for delivering and controlling the vibration from the generating devices to the plastic melt. Gas vibration can be applied at various stages and locations during processing and vibrated from subsonic, low frequency (1 to 30 Hertz) all the way up to ultrasonic range (15,000 to 20,000 Hertz). Whereas high-frequency vibrations fuse weld lines, accelerates the stress relaxation of plastics, and alter the rate of crystallization and crystal growth, low-frequency vibrations help change molecular orientation in a desirable way. In addition, resonance, which is a condition at which the entire system vibrates at its natural frequency, extracts the energy from the vibration and produces physical changes in the targeted system. One of the unique features of vibration gas injection molding is its ability to combine different modes of various frequencies and amplitudes to take advantages of their distinctive effects on the plastic melt.

The pressurized gas can be introduced directly into the mold from the back of the cavity, through the injection nozzle, or via channels or shaped chambers at specific locations in the mold near the runners and gates. Gas vibration is generated by the vibration sources, such as pneumatic, electrical, or mechanical transducers. Other hardware required by vibration gas injection molding includes a gas mixing unit, gas pressuring/injection units, and a computer control system for monitoring and maintaining resonant condition.

The unique advantage of this process is the ability to alter the morphology of the plastics, thereby modifying the mechanical properties, such as tensile strength, modulus, and impact. For example, it reportedly boosted the crystallinity in polypropylene by 25%, which resulted in improvement in low-temperature impact and clarity [119]. Detailed descriptions on the process benefits and drawbacks can be found in the various sections that deal with other melt-flow oscillation techniques.

9.21   Rubber Injection*

9.21.1   Rubber Molding Processes

Compression Molding

During compression molding, a piece of preformed material is placed directly in the mold cavity and compressed under hydraulic clamp pressure. At the completion of the required cure cycle, the hydraulic clamp is released and the part is stripped from the mold (cf. Fig. 9.66). Compression molding is usually done in hydraulically clamped presses acting through a ram.

As the mold cavities are open when the preform is introduced, compression molding usually possesses the least tolerance control and most flash. Exact weight is required in the preforming stage to ensure that additional variation is minimized. In

---

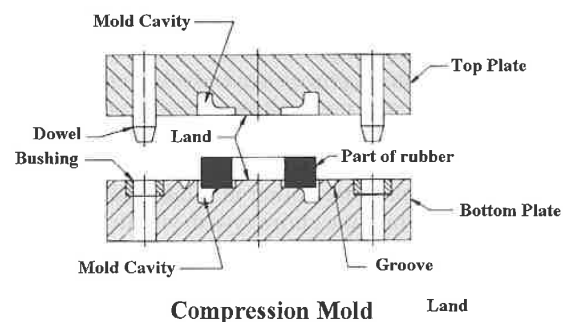* Contributed by Mauricio DeGreiff and Nelson Castaño.

Figure 9.66 Schematic of compression molding for rubbers.

addition, stock flow and air venting must be carefully addressed. Along with the preform shape, careful selection of the press closing rate and the bump cycle are required. The press bumping is a short pressure release after initial clamping to "burp" the mold.

As little shear is evident in the compression process, temperatures are obtained only by platen contact with the mold. Preheating of the rubber is occasionally used to speed the curing process. The dimensional requirements of the part most often determine whether or not to use compression methods.

Transfer Molding

Transfer molding is a precision molding technique. As shown in Fig. 9.67, the preformed stock flows from a transfer pot, usually above the mold cavities, to the parts below through sprues. The transfer ram and pot can be part of the mold or can be part of the press. All transfer molding is done with a closed cavity; therefore, less finishing is required. In addition, more precision is obtained. Transfer molding, however, is also a low shear process and little self-heating of the compound takes place. As a result, cure times are not much shorter than compression molding.
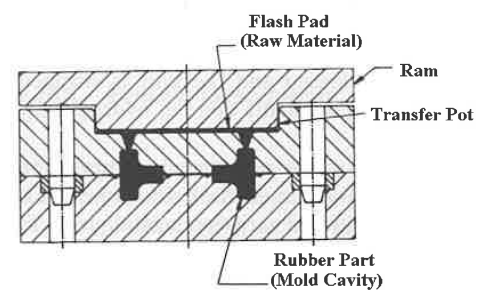
Figure 9.67 Schematic of transfer molding for rubbers.

Injection Molding

Injection molding is a versatile, high-precision process. The cost to produce a part is often the lowest, although the press and molds are the most expensive. The injection process employs a closed mold, with the stock delivered under high shear from an injection barrel (cf. Fig. 9.68). Several advantages result from this high shear. First, the rubber is at elevated temperatures as it enters the cavity, allowing for short cure times. This is most particularly advantageous in thick cross-section parts. Second, the viscosity of the rubber compound is lowered, allowing for considerable thermoplastic flow.
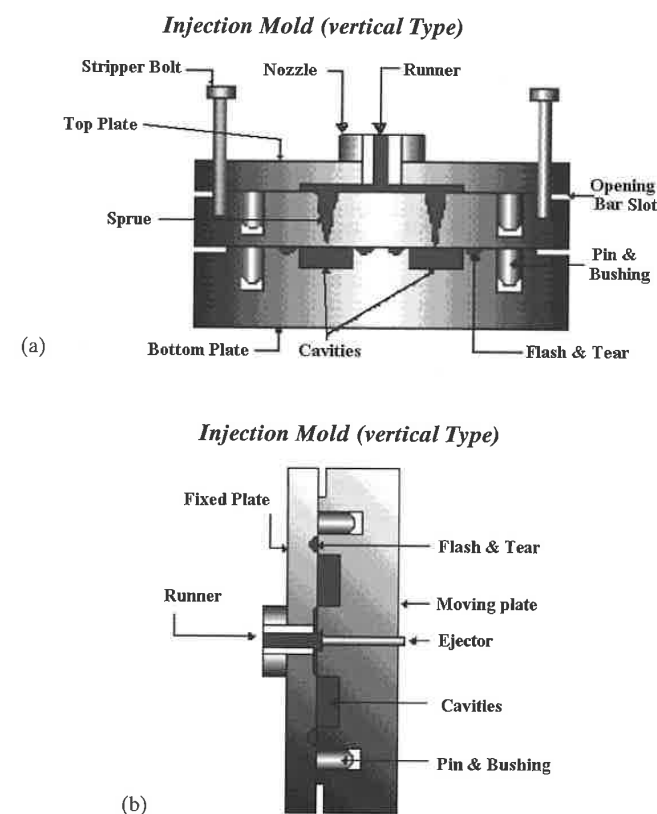
Figure 9.68 Schematic of injection molding for rubbers.

Injection presses can be vertical or horizontal. They can also use either a ram or a screw to masticate and deliver the uncured compound to the mold. The reciprocating screw masticates stock better and maintains even heat, but shot sizes are limited. The ram can deliver large quantities of stock to the mold, frequently at the expense of uniform mastication and heat. Hybrid processes have thus been developed, with a screw/ram combination commonly used. In these cases, the screw is used only for stock mastication, whereas the ram is used in injection.

Referring to the following figures, the uncured stock is delivered through an injection nozzle into a sprue bushing. Rubber flows through the runner system into the parts via sprues. It is most important that the flow of stock and heat history of the compound be as uniform as possible. Symmetrical or "balanced" runner systems are therefore employed.

As the mold is closed and the rubber heated during injection, tight tolerance control is achieved with minimal flash. If finishing is required, it is usually adequate to employ a bulk cryogenic tumbling technique.

A negative factor with injection molding is the large material waste generated through the runner system. "Cold runner systems" have been designed to address this deficiency. On the other hand, injection molding has many advantages. Tooling for this process, however, is most expensive because high-precision, hardened tools are often required for the high volumes encountered.

9.21.2   Curing Systems in Rubber Injection Process

Curing Systems in Common Use

A manufacturer transferring from a compression to an injection molding process may fairly safely carry out the first trials without modifyirig the compound, relying on adjustments of barrel temperature to obtain reasonable operating conditions.

Some typical formulations for NR and NBR polymers are listed in Tables 9.18 and 9.19, together with curing systems selected to offer a range of processing and cure requirements. These are based either on MBTS, sulphenamides, or Sulfasan R because of the need for a certain minimum of scorch safety in the compounds.

Efficient Vulcanizing Systems

Efficient vulcanizing (EV) systems are defined as those where a high proportion of sulphur is used for a cross-linking purpose. These systems have two main advantages over conventional systems, giving vulcanizates with reduced reversion and better aging characteristics. In addition to these advantages, EV systems based on dithiodimorpholine (DTM) are very versatile, enabling a wide range of scorch times, cure rates, and states of cure to be chosen at will.

It is particularly important for injection molding of thick sections to avoid reversion. EV systems give the complete answer to this problem. The conventional system (sulphur/MBTS/DPG) shows reversion immediately after the maximum modulus is reached, whereas the EV system (DTM/MBTS/TMTD) shows no reversion even

Table 9.18   Typical Formulations for NR Polymers

Black NR formulations					
Natural Rubber	70				
Whole tyre reclaim	60				
Carbon black	75				
Zinc oxide	40				
Stearic acid	5				
Paraffin wax	2				
Antiozonant	1				
Curing systems	A	B	C	D	E
Sulphur	2.5	2.5	2.5	—	—
Dithiodimorpholine—DTM	—	—	—	1.4	1.2
Sulphenamide	1.2	1.2	—	1.4	1.2
MOR	—	—	1.2	—	—
Thiuram	0.3	—	—	0.2	0.5

A, B, and C are suitable for thin-section products and are in ascending order of scorch time. D and E are efficient vulcanizing systems suitable for thick sections. They give much-reduced reversion and improved ageing resistance.

after three times the optimum cure time. EV systems can be developed to give equivalent cure properties with much improved aging as compared with a conventional cure, even when antioxidants are omitted.

Conclusions

Accelerator systems for injection molding should be chosen to give adequate scorch time, fast cure without reversion, and appropriate product properties. When molding

Table 9.19   Typical Formulations for NBR Polymers

Nitrile rubber formulations (NBR)				
Nitrile	100			
Carbon black	80			
Dioctyl phthalate	5			
Zinc oxide	5			
Stearic acid	1			
Curing systems	A	B	C	D
Sulphur	1.5	1.5	0.5	—
TMTD	0.5	—	3.0	—
MBTS	1.0	1.5	—	3.0

A and B are conventional curing systems that may be adequate where aging resistance is not a particular problem. C is a low sulphur system giving much-improved aging, but its scorch time is usually sufficient only for ram type injection. D combines excellent aging with a scorch time long enough for most applications.

thick sections from polymers that revert (e.g. NR), EV systems should be used to minimize reversion. Combinations of a sulphenamide, DTM, and TMTD are ideal, and the ratios can be varied to meet precise machine operating conditions and product requirements. Conventional sulphur/accclerator systems can be used where reversion is not a problem, and the following accelerators will give the best cure rates for each scorch time requirement.

Conventional sulphur/accclerator systems can be used where reversion is not a problem, and the accelerators, such as MOR, TBBS, and CBS/TMTD (in the order of decreasing scorch time), will give the best cure rates for each scorch time requirement. Accelerator loadings may be increased to give improved product properties or to counter the effect of oil addition.

Acknowledgments

References

1. Rosato, D. V., Rosato, D. V., *Injection Molding Handbook: The Complete Molding Operation Technology, Performance, Economics*, Second Ed. (1995), Chapman and Hall, New York.
2. Wang, K. K., et al., *Computer-Aided Injection Molding Systems, Progress Reports Nos. 1–21* (1975–2000) Cornell Injection Molding Program (CIMP), Cornell University, Ithaca, New York.
3. Kamal, M. R., Kenig, S., *Polym. Eng. Sci.* (1972) 12, 294, 302.
4. Berger, J. L., Gogos, C. G., *Polym. Eng. Sci.* (1973) 13, 102.
5. Tadmor, Z., *J. Applied Polym. Sci.* (1974) 18, 1752.
6. Hieber, C. A., Shen, S. F., *J. Non-Newtonian Fluid Mechanics* (1980) 7, 1.
7. Wang, V. W., Hieber, C. A., Wang, K. K., *J. Polym. Eng.* (1986) 7(1), 21.
8. Isayev, A. I. (Ed.), *Injection and Compression Molding Fundamentals* (1987) Marcel Dekker, New York.
9. Coyle, D. J., Blake, J. W., Macosko, C. W., *AICHE Journal* (1987) 33(7), 1168.
10. Manzione, L. T. (Ed.), *Applications of Computer Aided Engineering in Injection Molding* (1987) Hanser, Munich.
11. Lee, L. J., Wu, C. H., Chiu, H. T., Nakamura, S., *International Polymer Processing* (1998), 13(4), 389.
12. Tucker, C. L., *Fundamentals of Computer Modeling for Polymer Processing* (1989) Hanser, Munich.
13. Advani, S. G. and Beckermann, C. (Ed.), *Heat and Mass Transfer in Solidification Processing* (1991) HTD-Vol. 175/MD-Vol. 25, ASME, New York.

14. Castro, J., Conover, S., Wilkes, C., Han, K., Chiu, H. T., Lee, L. J., *Polymer Composites* (1997) 18(5), 585.
15. Wang, H. P., Turng, L. S., Marchal, J. M. (Eds.), *CAE and Intelligent Processing of Polymeric Materials* (1997), MD-Vol. 79, ASME, New York.
16. Turng, L. S., Wang, H. P., Ramani, K., Benard, A. (Eds.), *CAE and Related Innovations for Polymer Processing* (2000), MD-Vol. 90, ASME, New York.
17. Wang, J. T., In *CAE and Related Innovations for Polymer Processing*, Turng, L. S., Wang, H. P., Ramani, K., and Benard, A. (Eds.), (2000), ASME, New York, pp. 257–271.
18. Turng, L. S., Wang, V. W., Wang, K. K., *J. Eng. Mat. Technol.* (1993) 115, 48.
19. Turng, L. S., *Adv. Polymer Technol.* (1995) 14(1), 1.
20. Chen, S. C., Chen, Y. C., Cheng, N. T., Huang, M. S., *J. Reinforced Plastics and Composites* (1999), 18(8), 724.
21. Escales, E., *German Plastics* (1970) 60, 16–19. Translated from *Kunststoffe* (1970) 60, 847.
22. Eckardt, H., Davies, S., *Plastics and Rubber International* (1979) 4(2), 72.
23. Kirkland, C., *Injection Molding Magazine* (May 1996), 85.
24. Eckardt, H., *German Plastics* (1985) 75(3), 10–15. Translated from *Kunststoffe* (1985) 75(3), 145.
25. Avery, J. A., *Injection Molding Alternatives—A Guide for Designers and Product Engineers* (1998), Hanser, Munich.
26. Young, S. S., White, J. L., Clark, E. S., Oyanagi, Y., *Polym. Eng. Sci.*, (1980) 20(12), 798.
27. Avery, J. A., In *Innovation in Polymer Processing—Molding*, Stevenson, J. F. (Ed.) (1996), Hanser, Munich, 331–365.
28. Eckardt, H., Paper presented at the Fourteenth Annual Structural Foam Conference and Parts Competition, Boston, MA, April (1986).
29. Hagen, R., *German Plastics* (1989) 79(1), 27–30. Translated from *Kunststoffe* (1989) 79(1), 72.
30. Hauck, C., Schneiders, A., In *Innovation in Polymer Processing—Molding*, Stevenson, J. F. (Ed.) (1996), Hanser, Munich, 151–191.
31. Schmachtenberg, E., Polifke, M., *Kunststoffe Plast Europe* (1996) 86(3), 16. Translated from *Kunststoffe* (1996) 86(3), 322.
32. Gabriele, M. C., *Mod. Plast.* (April 1999), 70.
33. Eckardt, H., In *Innovation in Polymer Processing—Molding*, Stevenson, J. F. (Ed.) (1996) Hanser, Munich, 1.
34. Turng, L. S., In *Innovation in Polymer Processing—Molding*, Stevenson, J. F., (Ed.) (1996) Hanser, Munich, 43.
35. Hettinga, S., Product Brochure, Hettinga Technologies, Inc. Des Moines, IA, Web site address: *http://www.hettingatechnology.com/*
36. Michaeli, W., Brunswick, A., Kujat, C., *Kunststoffe Plast Europe*, (2000) 90(8), 25. Translated from *Kunststoffe* (2000) 90(8), 67.
37. Moore, S., *Mod. Plast.*, January (1993) 26.
38. Pearson, T., Product Brochure, Gas Injection Limited, Cheshire, U.K.
39. Moore, S., *Mod. Plast.*, November (1999) 28.
40. Kleba, I., Haberstroh, E., *SPE ANTEC Tech. Papers* (2000), 559.
41. Hopmann, C., Michaeli, W., *SPE ANTEC Tech. Papers* (1999), 417.
42. Potente, H., Hansen, M., *Int. Polym. Processing* (1993), 8(4), 345.
43. Dubois, J. H., Pribble, W. I., *Plastics Mold Engineering Handbook* (1987) Van Nostrand Reinhold Company, New York.
44. Bürkle, E., Wohlrab, W., *Kunststoffe* (1999) 89(9), 64.
45. Wang, J. T., In *CAE and Intelligent Processing of Polymeric Materials*, Wang, H. P. et al. (Eds.) (1997), ASME, New York, pp. 83–96.
46. Kim, I. H., Park, S. J., Chung, S. T., Kwon, T. H., *Polym. Eng. Sci.* (1999), 39(10), 1930, 1943.
47. Chen, S. C., Chen, Y. C., Peng, H. S., *J. of Applied Polym. Sci.* (2000), 75(13), 1640.
48. *Engineering Design Database Design Guide* (1991) GE Plastics, Pittsfield, MA, USA.
49. Pötsch, G., Michaeli, W., *Injection Molding—An Introduction*, (1995) Hanser, Munich.
50. Gabriele, M. C., *Mod. Plast.* April (1999), 76, 70.

51. Grande, J. A., *Mod. Plast.* September (1998) 75, 35.
52. Snyder, M. R., *Mod. Plast.* October (1999) 76, 54.
53. Jaeger, A., Fischbach, G., *Kunststoffe German Plastics* (1991) 81(10), 21. Translated from *Kunststoffe* (1991) 81(10), 869.
54. Bürkle, E., Rehm, G., Eyerer, P., *Kunststoffe Plast Europe* March (1996) 86, 7. Translated from *Kunststoffe* (1996) 86(3), 298.
55. Böcklein, M., Eckardt, H., *Kunststoffe German Plastics* (1986) 76(11), 17. Translated from *Kunststoffe* (1986) 76(11), 1028.
56. Steinbichler, G., *Kunststoffe Plast Europe* March (1995) 85, 18. Translated from *Kunststoffe* (1995) 85(3), 337.
57. Anders, S., Littek, W., Schneider, W., *Kunststoffe German Plastics* (1990) 80(9), 20. Translated from *Kunststoffe* (1990) 80(9), 997.
58. Shah, S., Kakarala, N., *Plastics Engineering* September (2000) 56, 51.
59. Snyder, M. R., *Mod. Plast.* November (1998) 75(11), 71.
60. Anonymous, *Kunststoffe German Plastics* (1978) 68(11), 4. Translated from *Kunststoffe* (1978) 68(7), 394.
61. Schrenk, W. J., U.S. Patent 5 202 074 (1993).
62. Barger, M. A., Schrenk, W. J., *In Innovation in Polymer Processing—Molding*, Stevenson, J. F. (Ed.) (1996) Hanser, Munich, pp. 293–330.
63. Hettinga, S., *In Innovation in Polymer Processing—Molding*, Stevenson, J. F. (Ed.) (1996) Hanser, Munich, 193.
64. Turng, L. S., Chiang, H. H., Stevenson, J. F., *Plastics Engineering* October (1995) 33.
65. Turng, L. S., Chiang, H. H., Stevenson, J. F., *SPE ANTEC Tech. Papers* (1995) 668.
66. Snyder, M. R., *Mod. Plast.*, January (1999) 76(1), 85.
67. Ruprecht, R., Bacher, W., Hausselt, J. H., Piotter, V., *Proc. SPIE Symp. Micromachining and Microfabrication Process Technology* (1995) 2639, 146.
68. Weber, L., et al. *Proc. SPIE Symp. Micromachining and Microfabrication Process Technology II* (1996) 2879, 156.
69. Weber, L., Ehrfeld, W., *Kunststoffe Plast Europe* (1998), 88(10), 60–63. Translated from *Kunststoffe* (1998) 88(10), 1791.
70. Kukla, C., Loibl, H., Detter, H., Hannenheim, W., *Kunststoffe Plast Europe* (1998) 88(9), 6–7. Translated from *Kunststoffe* (1998) 88(9), 1331.
71. Seidler, D., Zelenka, R., *Kunststoffe Plast Europe* (1998), 88(9), 7. Translated from *Kunststoffe* (1998) 88, 1338.
72. Weber, L., Ehrfeld, W., *Kunststoffe Plast Europe* (1999) 89(10), 64. Translated from *Kunststoffe* (1999) 89(10), 192.
73. Becker, E. W., et al. *Microelectron. Eng.* (1986) 4, 35.
74. Ehrfeld, W., et al., *J. Vac. Sci. Technol. B* (1988) 6, 178.
75. Michaeli, W., Rogalla, A., *Kunststoffe für die Mikrosystemtechnik* (1997) 6, 50.
76. Schinköthe, W., Walther, T., *Kunststoffe Plast Europe* (2000) 90(5), 17–19. Translated from *Kunststoffe* (2000) 90(5), 62.
77. Holzhauer, M., Zippmann, V., *Kunststoffe Plast Europe* (2000) 90(9), 18. Translated from *Kunststoffe* (2000) 90(9), 64.
78. Mapleston, P., *Mod. Plast.*, July (1997) 74, 41.
79. Suh, N. P., "Microcellular Plastics," In *Innovation in Polymer Processing—Molding*, Stevenson, J. F. (Ed.) (1996) Hanser Publishers, Munich, 93–149.
80. Park, C. B., Doroudiani, S., Kortschot, M. T., *Polymer Engineering and Science* (1998) 38(7), 1205.
81. Shimbo, M., Baldwin, D. F., Suh, N. P., *Polymer Engineering and Science* (1995) 35(17), 1387.
82. Seeler, K. A., Kumar, V., *Journal of Reinforced Plastics and Composites* (1993) 12, 359.
83. Gulari, E., Manke, C. W., Paper presented at the *219th ACS National Meeting, I&EC*—106, San Francisco, CA, March 26–30 (2000).
84. Royer, J. R., Gay, Y. J., Desimone, J. M., Khan, S. A., *Journal of Polymer Science, Part B: Polymer Physics* (2000) 38, 3168.

85. Kwag, C., Manke, C. W., Gulari, E., *Journal of Polymer Science, Part B: Polymer Physics* (1999) 37, 2771.
86. Zhang, Z., Handa, Y. P., *J. Polym. Sci. Part B: Polym. Phys.* (1998) 36, 977.
87. Handa, Y. P., Kruus, P., O'Neill, M., *J. Polym. Sci., Part B: Polym. Phys.* (1996) 34, 2635.
88. Knights, M., *Plast. Tech.* (Sept. 2000) 46, 40.
89. Trexel, Inc. Web site, *http://www.trexel.com/*.
90. Malloy, R. A., *Plastic Part Design for Injection Molding—An Introduction* (1994), Hanser Publishers, Munich.
91. Mapleston, P., *Mod. Plast.* April (2000), 46–51.
92. Product brochure, *The SCORIM Process*, SCORTEC, Inc., Gulph Mills, PA.
93. Reis, R. L., Cunha, A. M., Bevis, M. J., *Mod. Plast.* May (1999) 76(5), 73.
94. Hills, K. Paper presented at Molding '95. March (1995), New Orleans, LA.
95. Rawson, K. W., Allan, P. S., Bevis, M. J., *Polym. Eng. Sci.* (1999) 39(1), 177.
96. Becker, H., Malterdingen, G. F., Müller, U., *Kunststoffe German Plast* (1993) 83(3), 3. Translated from *Kunststoffe* (1993) 83(3), 165.
97. Theberge, J., *Plast. Eng.* February (1991) 47, 27.
98. Becker, H., Ekenhorst, D., Fischer, N., Paper presented at the *Int. Symp. on Advanced Materials for Lightweight Structures*, March (1994), ESTEC, Noordwijk.
99. Harada, T., *Kogyo Zairyo (Engineering Materials)* (1991), 39(15), 65 (in Japanese).
100. German, R. M., Bose, A., *Injection Molding of Metals and Ceramics* (1997), Metal Powder Industries Federation, Princeton, NJ.
101. Ballard, C., Zedalis, M., *SPE ANTEC Tech. Papers* (1998), 358.
102. Bayer, M., *Kunststoffe Plast Europe* (2000), 90(7), 17. Translated from *Kunststoffe* (2000) 90(7), 46.
103. Schumacher, C., *Kunststoffe Plast Europe* (1999), 89(9), 23. Translated from *Kunststoffe* (1999) 89(9), 70–72.
104. Petzoldt, F., *Kunststoffe Plast Europe* (2000), 90(7), 14–16. Translated from *Kunststoffe* (2000) 90(7), 40–43.
105. Johnson, C. F., *Engineering Plastics*, Vol. 2, Engineered Materials Handbook, ASM International (1988) 344.
106. González-Romero, V. M., Macosko, C. W., *Polym. Eng. Sci.* (1990) 30(3), 142.
107. Ibar, J. P., *Plast. Eng. Sci.*, (1998) 38(1), 1–20.
108. Ibar, J. P., *Mod. Plast.* January (1995) 72(1).
109. Ibar, J. P., U.S. Patent 5,605,707 (1995).
110. Brewer, G. W., In *Engineered Materials Handbook* (1998) Vol. 2, ASM International, 508.
111. Wu, J. S., Lee, M. J., *Plastics, Rubber and Composites Processing and Applications* (1994) 21(3), 163.
112. Tewald, A., *Kunststoffe Plast Europe*, July (1998) 88(7), 13. Translated from *Kunststoffe* (1998) 88(7), 970.
113. Macosko, C. W., *RIM Fundamentals of Reaction Injection Molding—Introduction* (1988), Hanser, Munich.
114. Farrah, M., Hanson, D., Wendle, B., (Eds.), *Structural Foam* (1984), Structural Foam Division, Society of Plastics Industry, Washington, DC, USA.
115. Fassett, J., *Plastics Engineering*, December (1995) 35–37.
116. Stevenson, J. F., Dubin, A., *J. Injection Molding Technology*, December (1999) 3(4), 181.
117. GE Plastics, Thin-wall Technology Guide (1998).
118. Wang, H. P., Ramaswamy, S., Dris, I., Perry, E. M., Gao, D., In *CAE and Related Innovations for Polymer Processing*, Turng, L. S., Wang, H. P., Ramani, K., Benard, A., (Eds.) (2000), MD-Vol. 79, AMSE, New York, 193.
119. Ibar, J. P., *SPE ANTEC Tech. Papers* (1999), 552.
120. Ibar, J. P., U.S. Patent 5,705,201 (1998).

10 Part Design

J. Beaumont

10.1 The Design Process

The successful design of a plastic part is one of the greater challenges and financial risks an engineer or a company has to face. Before a plastic part can be truly evaluated, a mold must be custom designed and built that can easily cost tens to hundreds of thousands of dollars. This can be several million times the selling price of the part being designed. Furthermore, the process of designing, building a mold, and molding the first plastic part can easily take 20 weeks. Finally, after molding the parts in the new mold, the parts can be evaluated for size, shape, and mechanical properties. It is rare that these first parts are to the required specifications. The next stage is typically a long, costly process of trying to produce parts to specification. This commonly includes changes to the part design and/or the plastic material, and, therefore, to the mold design and process.

The problem with successfully producing plastic parts arises from the complex nature of plastic materials and how their characteristics can be significantly influenced by the part design, tooling, and molding process. This problem is accentuated by the fact that a majority of plastic parts are designed with only a minimum consideration of the molding requirements, and a minimum appreciation of the significant impact of these requirements on the molded part. As a result, it is common that a mold is built and then modified numerous times in order to produce the required product.

After years of numbing experience, many companies have adopted a process of rushing through the part design, and then to have the mold designed and built in order to shoot parts just to find out what is wrong and begin the debugging process. This process continues today despite the availability of new computer aided engineering (CAE) technologies, such as injection molding simulations and structural analysis, which can reduce many of the risks. The perception is that seeing a mold being built is making progress, whereas up-front design engineering increases the time to produce a product. The additional time to debug the design of the mold with the lost potential of an optimized product is considered part of the business. One of the reasons for the limited acceptance of up-front engineering is that it is less tangible and more abstract. It is difficult to determine what should be done, how much is required, what tools are required, how good the tools are, how efficiently they are being used, and how to measure their actual contribution to the final product.

Each product presents a different degree of challenge. A judgment must be made as to what level of up-front engineering should be applied versus the risk. The more time invested in engineering, the lower the risk; however, there is a diminishing return on the use of the up-front engineering time. If left alone, an engineer could engineer forever. The product may never make it to market if the engineer is expected to design a product guaranteed to succeed.

The successful design of a plastic part requires numerous activities. Some of these are common to that of any new product or design. A product concept is developed by someone with the intention of making a profit. To get from the concept phase to the money-making phase, a number of tasks are required of numerous different individuals. Some of these tasks must be performed sequentially, whereas others are best performed concurrently. The progression of sequential tasks is not always the same. For example, some material requirements dictate a design, whereas some designs dictate the material. The process of selecting a material, therefore, may come before, after, or concurrently with the part design. Some tasks need more attention in some projects than others; regardless, the paramount factors on which the success of the project depends are to identify clearly the objective of the project, to plan, and then to communicate these objectives to all parties involved.

The design process presented in the following provides a list of tasks in the order that they might be conducted. This sequence may vary from project to project. Following data collection, and the development of a project plan, many of the tasks are best performed concurrently to minimize problems with product performance and production.

1. Data collection and product specifications
   - Collect information on the product and customer
   - Start a folder for information as soon as it comes in
   - Develop product specifications based on the collected data
   - Determine a beginning and end date for the project
   - Once all data has been collected, product specification written, and timetable established, have your customer review and sign off
   - Do *not* begin designing a product until all parties agree on the product specifications and timetable
2. Project plan
   - A project plan should be made in the early stages of the project. A preliminary plan may be made before *data collection* and then be altered and developed based on any newfound information. It can be expected that the plan will be altered throughout the project, but it provides an important guideline to accomplish your design and time objectives.
   - The project plan may be very basic or quite elaborate, depending on the complexity of the project. At a minimum a project plan should:
     - Establish a *timetable* for project *tasks*
     - Allocate *resources*. Resources can include personnel, equipment, technology, money, and so on

   - Without a project plan you significantly increase your risk of failure to meet your obligations

It is critical to meet both design objectives as well as time objectives.

   - Establish project deadline
   - Identify resources (including personnel)
   - Identify tasks
   - Assign resources to tasks
   - Develop schedule for resources and tasks (check for overlap)
   - Determine the critical path
   - Order long delivery items or information first

3. Preliminary Design
   - Develop sketches of a variety of design options
   - Be creative. Attempt to isolate yourself from previous designs in order not to inhibit creativity and originality. If a design team exists, develop initial concepts individually.
   - After initial concept designs, review with team members and against existing products. Channel and merge ideas progressively until an optimum design is established.
   - Challenge your design and the design of team members. Do not become overly attached to a given design that might blind you from new and better ideas
   - Review viability of the various options that consider manufacturability, cost, and risk
   - Can components in the design be merged, or can value be added, because of inherent material or process capabilities of plastics and the injection molding process? (This is commonly referred to as "Value Engineering".) Speculate on alternate approaches to the design.
   - Isolate weaknesses and assumptions that might compromise the final product. Review these and further develop them.
   - Anticipate part requirements (structural, environment, cosmetic, etc.)
   - Refine sketches and develop mock-ups (these may be done with cardboard, foam, CAD, etc.)
   - Do not be limited by things you do not know. Seek assistance.
   - Never design too much at a given session. Break it into as many discrete pieces as possible, avoiding fatigue, but preserving continuity.
   - Perform preliminary analyses where appropriate. This should be simplified work that might include hand calculations, use of spreadsheets, and mold filling analysis.
4. Material Selection: An initial material selection must be made before the detailed design can begin. In some case the further design of the part might dictate that

the material be changed. Material selection is presented in more detail later in this chapter.

5. Develop Detailed Design
   - *Function*: Gather the design team and interested parties into a small group and narrow concepts, addressing functions, into a single design. Make critical decisions.
   - Apply design guidelines as specific to the selected plastic materials and processes.
   - *Structural*: Perform appropriate structural analyses through calculations or structural analysis software. Apply appropriate safety factors.
   - *Process*: Review process effects as related to the design. This might relate to wall thicknesses, tooling requirements, and so on.
   - Develop detailed part drawings suitable for production or prototyping.
   - Simple solutions are always preferred.

6. Testing/Prototyping
   - It may be appropriate to test or prototype your design, or components of your design, before committing to production. Prototypes can serve various functions, including evaluation of fit, form, function, and structural considerations. For structural and fit evaluation it is often required that the part be produced from the same material and by the same process that will be used in production. Other than for structure and fit, evaluation may often be serviced with alternate materials and may be produced through means other than those to be used for the production product. Techniques that are common include:
     - Machining of materials (wood, plastic)
     - Stereolithography or alternate rapid prototyping methods
     - Casting of urethane parts from silicone molds produced with a master
       - Masters can be machined or formed through stereolithography or an alternate rapid prototyping method
     - CAD and simulation software
   - With plastic products there may be particular concern with the final size, shape, and mechanical properties due to the influence of the process. This has classically been addressed by producing parts that employ prototype tooling. Caution must be maintained because differences between the prototype tool and the production tool will create variations in the prototype part. One must clearly ascertain the objectives of a prototype mold and assess what is required of the tool to achieve these objectives. Variations in mold steel, cooling, gating location, number of cavities, and the like, will create differences between the parts molded in the prototype versus the production mold.
     - Many advancements in CAE and material characterization techniques are improving the ability to evaluate a part without physical prototyping.

7. Review design and revise through Steps 4 to 6
   - Review all assumptions
   - Review and critique any simulation work closely
   - Try to view design with a new critical eye
   - Avoid design momentum—do not stick with a design that is considered flawed

8. Commit to the design and develop project plan to bring to production
   - Avoid running out of resource
   - Avoid simultaneous efforts and over commitment
   - Establish critical path, flags, and goals—both final and intermediate

10.2   The Four Building Blocks of Plastics Part Design

The successful development of injection molded plastic parts is significantly handicapped by the complex interaction between plastics materials, part design, mold design, and processing. These four building blocks each affect how the part is designed, and how it will behave once it is produced. Each of these four factors individually present a significant challenge. When one must consider their interrelation in the successful development of a plastic product, the task can seem overwhelming; however, new plastic parts are developed every day.

Due to the complexity of developing an injection molded part, it is best that the development be performed with the cooperative effort of individuals with expertise in each of the four major areas. By involving a process engineer and a mold designer during the early stages of part design, many manufacturing problems might be avoided. Technologies such as structural analysis and injection molding simulation programs can also act as a significant aid in product development. With the use of injection molding simulation, a skilled analyst can evaluate many of the manufacturability issues as the design is evolving. Without a good understanding of this technology, however, a designer can develop false confidence. It should be recognized that these programs provide information that requires skilled interpretation.

The following sections discuss each of the four key building blocks of developing a plastic product and some of the complexity that they contribute.

10.2.1   Material

More than 100,000 variations of plastic materials are available. This creates significant challenges for a designer to sort through and determine which provides the combination of optimum performance and manufacturability at a minimum cost. In addition to screening material requirements based on part application, one must consider whether the material can be molded into the geometry of a physical part design. The physical part design must satisfy the mechanical requirements of the part, and have the ability to be produced in, and ejected from, a mold.

The environments in which the part is to live must be considered when selecting a material for a given application. These requirements are often factors that cannot be influenced by the physical design of the plastic part. Material properties that cannot be affected practically by design include chemical resistance, UV resistance, colorability, transparency, HDT, and low shrink. For a given product application these

would be characterized as material requirements and would be a high priority in the selection process. Material properties such as modulus, impact strength, and tensile strength are mechanical properties and have a lower priority when selecting a material. Mechanical properties of a product are affected by both the product design and the material; therefore, selecting a material based on mechanical properties might limit the opportunities when designing a new part.

In addition, the characteristics of plastic materials are much more complex than are most other materials. Few of their properties are constant, and often only limited information on them is available. These nonconstant properties include fundamental characteristics such as modulus, yield strength, and viscosity. These can be affected by strain, strain rate, temperature, humidity, time, and process. Figure 10.1 illustrates the effect of temperature and strain rate on the stress–strain relationship of a plastic material. If a designer requires a materials modulus with which to design, the conditions under which the data was derived must be considered as to its relevance to his or her particular application. The standard published modulus for a material derived using ASTM test procedures has limited practical use in designing an under-hood automotive part. The ASTM procedure specifies that the plastic specimen be prepared at room temperature and at a relative humidity of 50%. The conditions under the hood of an automobile are quite different from this.

Figure 10.2 contrasts the stress–strain relationship of a metal versus a plastic. Note the lack of a significant linear region from which the modulus of the plastic material can be derived. When designing with metal, it is relatively easy to derive a modulus because of its strong linear region. In addition, the metal is generally used in low strain applications where the modulus is constant. With plastics, the modulus is nonlinear except at very small strains. This creates a problem because plastics are commonly used in applications that experience relatively high strains. The result is that the modulus is actually changing as the part is being distorted. Figure 10.3 illustrates the existance of six different moduli derived for a given plastic dependent on deflection. This indicates that as a component is deflecting, its modulus is constantly changing as the deflection progresses (see Fig. 10.4).
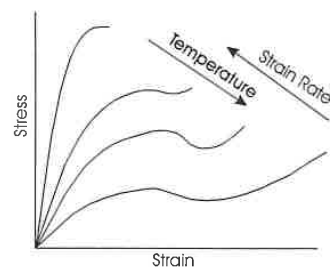
Figure 10.1 The effects of temperature and strain rate on the stress-strain relationship of a plastic material.
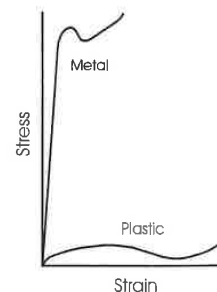
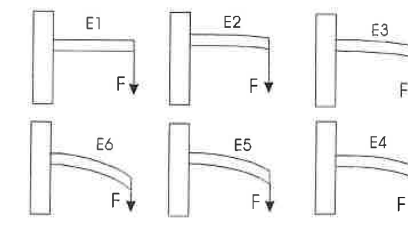Figure 10.2 Comparison of a typical stress-strain curve between a metal and plastic.
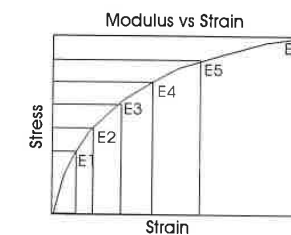
Figure 10.3 Six different moduli derived for the same plastic.

$$\text{Deflection} = FL^3/3EI$$

Figure 10.4 Modulus is constantly changing as the deflection progresses.

Plastic material properties also change with time. A part under load may initially experience very little strain. With time, under the same load, the part will continue to distort. This is commonly referred to as "creep."

Properties of plastics are also significantly affected by process. Factors such as cooling rate, flow direction, and flow type can all influence the orientation of the polymer molecule and any asymmetric fillers or reinforcements within the polymer. The orientation will create anisotropic properties in a molded part. ASTM tests do not specify how a specimen is to be prepared. The result is that most test specimens are molded at ideal conditions with the most favorable orientation. A designer must again be cautious of using this data and question its relationship to the part being designed.

Material Selection

With the wide variety of plastic materials, it is important to develop some methodology in identifying the optimum material for a given application.

1. Develop lists of the materials specifications. The first list should be *material requirements*. Requirements would include material properties that cannot be effected by varying the physical design of the part. These properties might include:
   - chemical resistance
   - colorability
   - clarity
   - thermal resistance

   The second list should be *material preferences*. Preferences are material properties that appear to fit the application, but which can be affected by part design. These are commonly mechanical properties. Though the part may require good rigidity, this can also be influenced by the design of the part; therefore, it is not mandatory that the material be a certain modulus for the part to be rigid. Other material preferences might include:
   - impact strength
   - tensile strength
   - modulus

2. Material selection should consider:
   - Mechanical properties
   - Ability to survive its environment
   - Wear
   - How the part is to be produced
   - Specialty properties/requirements (optical, lubricity, etc.)
   - Cost. In structural applications, cost should consider rigidity and strength of the material rather than just cost per volume. (Material cost issues are presented later in this chapter)
3. List the materials that satisfy your requirements and have some of your preferences. Next, rank these materials by cost and availability. Finally, rate each material for *processability*.
4. If your material selection is still not clear, pick the one with which your company is most familiar.
5. If a structural, or injection molding, analysis is to be performed, then the appropriate material data should be collected. In some cases the availability of this data may be considered in your material selection process, i.e., if you cannot analyze it, your design dapabilities will be limited.

Material Cost Considerations

After identifying candidate materials that satisfy requirements and preference, it is important to consider their cost relative to the parts performance. Material costs are generally provided in a cost per weight relationship ($/pound). Cost per weight does not always provide a good indicator as to the materials actual cost relative to your part application requirements. The effect of material density, strength, and rigidity on cost must be evaluated. Your part may require high strength, which would indicate that you need a high strength material; however, part strength can be affected by its structural design. By establishing a required modulus too early, therefore, you may unnecessarily limit your final selection and compromise your design.

Consider a structural part where both HDPE and a glass-filled nylon satisfy the material requirements. The cost per pound of the HDPE is considerably less than that of the nylon. Because the nylon has a much higher modulus, however, the part can be designed with a much thinner wall than if the HDPE is used. The thinner wall will reduce the amount of material used and the cooling time required. Both of these will reduce the cost of the molded part. Therefore, it is not clear which of these two materials is less expensive to use with the part and thus must be evaluated.

At a minimum, material costs should be contrasted based on volume rather than weight. In a structural application, the relative required wall thickness requirements of the candidate materials can also be contrasted. If the strength of a part is being considered, it should be realized that a part made from a Nylon could have a thinner wall than if it were produced from an HDPE. This will reduce the amount of material required and molding cycle time, thereby possibly reducing total part cost despite the higher cost per pound for the Nylon.

Although contrasting the structure of two parts made with different materials is best evaluated using structural analysis programs, the following method provides a

simplified means of determining equivalent cost of plastic materials based on structural requirements. Candidate materials can be evaluated based on their relative strength, as indicated by their *yield strength*, or for their rigidity, as indicated by their relative *moduli*.

Contrasting the Cost of Plastic Materials Based on Strength   This process provides a means to contrast the cost of plastic materials based on their ability to withstand stress.

1. Determine the primary type of stress the part will experience (tensile, compressive, flexural, shear, torsion).
2. Find the corresponding *yield stress* values of your candidate materials.
3. Determine the *yield strength ratio* (σ^*) of the candidate materials utilizing one of the materials as a reference material. In this example, there are two candidate materials: A" and "B."

$$\text{Yield Stress of Material "A"} = 10{,}000\,psi \quad \text{(Reference Material)}$$

$$\text{Yield Stress of Material "B"} = 7000\,psi$$

$$\sigma^* \ (\text{Yield Strength Ratio of "B" relative to "A"}) = \frac{7000}{10{,}000} = 0.7$$

4. Realizing that material "B" has only 70% of the strength of material "A," an equivalent geometry can be derived that would give a structure the same strength, whether it was made from material "A" or "B." To accomplish this, a formula is derived from the stress formula for the particular loading case for which your part will experience in application. In addition, the designer must consider which geometrical dimension in the part is considered a variable. For example, in a beam that will be exposed to a flexural load, is the width of the beam (wall thickness) or its height a variable?

In a flexural loading case, the following standard stress formula would be used.

$$\sigma = \frac{Mc}{I}$$

In a tensile or compressive loading case, the following simple relationship can be used.

$$\sigma = \frac{F}{A}$$

Example 10.1
Find the equivalent geometry for a *flexural load case* where part height (h) is the variable (i.e., what height is required of a part made from material "B" so

that it has the same strength of a part made from material "A"). As we are only looking for equivalence, one does not need to know the actual load conditions, nor the actual part dimensions. Regardless of the part's actual shape, the materials are contrasted assuming the structure is a simple beam with a height (h) and a width (w).

$$\sigma = \frac{Mc}{I} = \frac{M.5d}{wh^3/12} = \frac{6M}{wh^2} \quad d = \sqrt{\frac{6M}{w\sigma}}$$

where M = bending moment
$\quad c$ = distance from the neutral axis to outer surface
$\quad I$ = moment of Inertia

Because we are looking to contrast candidate materials under identical structural loads, and part width is not a variable, then

$$\frac{6M}{w}$$

is the same for either material and can be canceled from the equation; therefore, finding the equivalent height can be reduced to

$$h* = \sqrt{\frac{1}{\sigma*}}$$

Find the *equivalent height* ($h*$) of material "B" relative to material "A"

$$h* = \sqrt{\frac{1}{\sigma*}} = \sqrt{\frac{1}{.7}} = 1.195h$$

This indicates that a part that is produced from material having a yield stress of 7000 psi must be 1.195 time taller than a part being produced from the reference material that has a yield stress of 10,000 psi.

5. Find the current cost/volume ($/in³) of the material from the material suppliers. This will most likely require that cost be converted from cost per weight ($/pound) to cost per volume ($/in³).
6. Find the *equivalent cost/volume* of the candidate materials based on their relative heights and their relative costs per volume.

Equivalent Cost/Volume = ($/in³)(h*)

Contrasting the Cost of Plastic Materials Based on Rigidity   The following method provides a means to contrast the cost of plastic materials based on their ability to resist deflection.

Example 10.2
1. Determine the primary type of load the part will experience in application (flexural, tensile, compression, shear).
2. Find the corresponding *modulus* values for the candidate materials.
3. Determine the *modulus ratio* ($E*$) of the candidate materials using one of the materials as a reference. In this example, there are two candidate materials: "A" and "B."

Modulus of material "A" = 350,000 psi   (Reference Material)

Modulus of material "B" = 210,000 psi

Modulus Ratio of Material "B" relative to "A" = $E* = \dfrac{210,000}{355,000} = 0.592$

4. Realizing that material "B" has only 59.2% of the rigidity of material "A," an equivalent geometry can be derived that would give a structure with the same rigidity, whether it was made from material "A" or "B." To accomplish this, a formula is derived from the deflection formula for the particular loading case for which your part will experience in application. Again the designer must consider which geometrical dimension in the part is considered a variable (height or width).

   In a flexural loading case, any standard beam equation under a flexural load can be reduced to:

$$y = \frac{1}{EI} = \frac{1}{E\left(\dfrac{wh^3}{12}\right)}$$

   Again, because we are looking to find the relative height ($h*$) for a part made of material "B," which will provide the same rigidity as when produced from material "A," y and w will cancel out and be reduced to:

$$h* = 3\sqrt{\frac{1}{E*}}$$

5. Find the cost/volume ($/in³)
6. Find the "equivalent cost/volume" based on performance

$$= (\$/in^3)(h*)$$

Sample Problems

Three materials have been identified that satisfy the requirements of a part to be designed. These materials have the following properties. The part will be under a flexural load during application.

Material	Cost ($/lb)	Flexural Modulus (psi)	Flexural Yield Strength (psi)	Density (g/cm³)
A	0.60	200,000	8,000	1.25
B	0.52	105,000	7,500	1.05
C	0.75	260,000	10,000	0.98

** **Given** the following calculations for a rectangular beam under a flexural load.

$$\sigma = \frac{FLc}{I} \quad y = \frac{FL^3}{3EI} \quad I = \frac{wh^3}{I}$$

Problem 10.1

You are selecting a material for a part which will be under a flexural load during application. The height (h) of the part can be changed (is the variable). Determine which of these materials is the most cost effective based on rigidity.

1.1. Determine the Cost per Volume of the materials ($/in³)
1.2. Determine the ratio of moduli of the three materials (use material "A" as the reference material)
1.3. Derive the formula for the equivalent geometry
1.4. Find the equivalent geometry (height)
1.5. Find the equivalent cost

Answers Problem 10.1

1.1. "A" = $0.0271   "B" = $0.0197   "C" = $0.0266
1.2. "A" = 1   "B" = 0.525   "C" = 1.3
1.3. $h* = 3\sqrt{\dfrac{1}{E*}}$
1.4. "A" = 1   "B" = 1.2396   "C" = 0.9163
1.5. "A" = 0.0271   "B" = 0.0244   "C" = 0.0243

In this example, material "C," which is the most expensive material based on weight, is the least expensive material to use in application.

Problem 10.2

Determine which of the preceding materials is the most cost effective based on strength.

1.1. Determine the ratio of the yield strength of the three materials (use material "A" as the reference material)
1.2. Derive the formula for the equivalent geometry
1.3. Find the equivalent geometry (height)
1.4. Find the equivalent cost

Answers Problem 10.2

1.1. "A" = 1   "B" = 0.9375   "C" = 1.25
1.2. $h* = \sqrt{\dfrac{1}{\sigma *}}$
1.3. "A" = 1   "B" = 1.0328   "C" = 0.8944
1.4. "A" = 0.0271   "B" = 0.0204   "C" = 0.0237

In this case, material "B" is the least expensive material in application and in cost per pound. However its cost advantage over material "C" has been decreased.

Problem 10.3

Derive the formula for equivalent geometry if the primary load is flexural and the height of your part is fixed (therefore, only width w can be changed).

Answer Problem 10.3

- For Sample Problem 10.1 (solving for rigidity)

$$w* = \frac{1}{E*}$$

- For Sample Problem 10.2 (solving for strength)

$$w* = \frac{1}{\sigma *}$$

Problem 10.4

Derive the formula for equivalent geometry if the primary load in application will be compressive and you are concerned with the strength of the part.

Answer Problem 10.4

$$A* = \frac{1}{\sigma *}$$

Material Selection Effect on Cycle Time

To refine material cost impact on a molded part further, the effect of wall thickness on cycle time must be considered. Cycle time consists of fill, packing, cooling, and mold open (ejection) time. A thinner wall will normally reduce the filling, packing, and cooling time of a molded part.

Consider a structural case of a plastic shelf. The rigidity of the shelf is to be gained primarily from a series of parallel ribs spanning the length of the shelf. The height of the ribs are fixed at 1.5 in. Rib width (w) is considered the only variable. The two candidate materials that satisfy the requirements, and are being considered, for this application are HDPE and nylon. The cost of the nylon is more than three times that of the HDPE when contrasted based on dollars per pound ($1.50/lb vs. $0.43/lb). If you consider the relative modulus of the materials, the equivalent width of the ribs if produced from the Nylon is only 31% than if produced from HDPE. If you consider density difference the equivalent cost of the Nylon is now only 1.28 times that of the HDPE. Even though the Nylon still appears to be more expensive than the HDPE, the difference is significantly less than the original 3.49 cost ratio.

For full evaluation, however, the effect on production cost should be considered. A structural analysis determines that a 0.200-in rib thickness is required if the HDPE is to be used. Because the nylon part can be only 31% of this width to get the equivalent rigidity, the nylon part's wall thickness would be only 0.062 in thick. Cooling time for the 0.200-in-thick HDPE part is calculated to be approximately 26 seconds, whereas the Nylon is expected to cool in less than 3 seconds. Fill, pack, cooling, and mold open time are estimated to be 50 seconds for the HDPE and 13 seconds for the nylon part. Given a single cavity mold with production cost of $50/hour, one direct laborer at $10/hour and 5% spoilage, the production cost for the HDPE part is $ 0.88/part, whereas the Nylon part is only $0.23/part. Thus, the production cost for the HDPE part is nearly four times that of the Nylon part.

Relative cooling times of molded parts, as affected by material and wall thickness, can be approximated using the following formulas. Cooling time (t) is calculated based on wall thickness (h), thermal diffusivity (α), melt temperature (T_m), mold wall temperature (T_w), and ejection temperature (T_e). For solid round sections, as in bosses and runners, the radius (R) is used.

Two formulas are given for calculating approximate cooling time of both a flat plate and a round section. The first (t_c) calculates the time for the centerline of the plastic part to reach ejection temperature. The second (t_a) calculates the time for the average of the cross-section of the plastic part to reach ejection temperature.

Flat Plate

$$t_c = \frac{h^2}{\alpha\pi^2}\ln\left[\frac{4}{\pi}\left(\frac{T_m - T_w}{T_e - T_w}\right)\right]$$

Note: T_e = the desired temperature of the centerline of the part at ejection.

$$t_a = \frac{h^2}{\alpha\pi^2}\ln\left[\frac{8}{\pi^2}\left(\frac{T_m - T_w}{T_e - T_w}\right)\right]$$

Note: T_e = the desired temperature for the average part temperature at ejection.

Cylinder

$$t_c = 0.173\frac{R^2}{\alpha}\ln\left[1.6023\left(\frac{T_m - T_w}{T_e - T_w}\right)\right]$$

Note: T_e = the desired temperature of the centerline of the part at ejection.

$$t_a = 0.173\frac{R^2}{\alpha}\ln\left[0.6916\left(\frac{T_m - T_w}{T_e - T_w}\right)\right]$$

Note: T_e = the desired temperature for the average part temperature at ejection.

It is nearly impossible to predict an actual cooling time of a molded part accurately. Cooling time is only partly related to the time it takes for the plastic to reach a given temperature. Uniformity of cooling, melt flow patterns in a part, and material will affect shrinkage and the development of residual stresses that will tend to warp a part. Cooling time will also affect these and therefore becomes a process variable, which is difficult to determine until the part is molded. In addition, the shape of the part can have a significant effect. A flat part will have less structure to resist residual stresses and therefore might have to be constrained in a mold longer than a cylindrical part, which has a better structure to resist warpage. The longer cooling time will both lower the part wall temperature and thereby increase its rigidity at the time of ejection and increase the amount of time that the mold can act as a fixture, helping to control the parts final size and shape. Because these factors are not considered in the preceding calculations for cooling time, they should be used with caution when relying on them to estimate cycle times.

10.2.2 Product Design

The designer must design a product to satisfy its functional, structural, aesthetic, cost, and manufacturing requirements. These variables are often at odds with each other, and judgments must be made in balancing the conflicting requirements. An example might be a rib that is required for added structure or stability to reduce the potential for warpage. This rib, however, can create a sink on the primary show surface due to the high localized shrinkage resulting at the intersection of the rib and the primary wall. Widely accepted guidelines indicate that the rib should be thinned relative to the primary wall to reduce the large volume of material causing this localized shrinkage [see Fig. 10.5(a)].

This thinned rib, however, now presents a restricted channel that may result in mold filling problems during molding. The height of the rib, therefore, becomes limited which requires that multiple shorter thinned ribs be used to replace a single rib. Regardless of these approaches, the thinned ribs will shrink differently than will
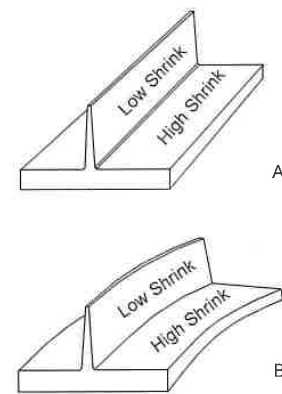
Figure 10.5 (A) Thinned rib to reduce localized shrinkage. (B) Warpage caused from variation in shrinkage between thin and thick regions.
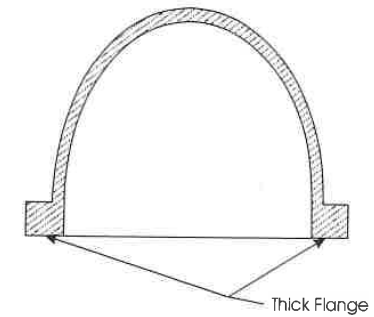

Figure 10.6 Cross section illustrating common flange/ primary wall relationship responsible for molding and warpage problems common in production.

the thicker primary wall. This variation in shrinkage will result in residual stress and often warp the part [Fig. 10.5(b)].

One of the significant advantages of plastic parts is that a part can be molded that incorporates a multitude of features that might otherwise require machining and assembly of multiple parts. The expectations of the plastic part and the challenge to the designer to satisfy the multiple functions therefore present further challenges. Compounding this challenge is the need to combine these features while not overly complicating the tooling requirements.

Radiator end tanks are a good example. These parts include an oversized flange (thicker than the primary wall) for mounting and sealing the tank to the radiator, a number of coolant hose connections, and a multitude of miscellaneous mounting brackets, widgets, and thingamajigs. These designs are produced for nearly every automotive company; however, the nature of the design puts a significant burden on the mold designer and molder. Despite years of experience with these tanks, a molder still must go through numerous revisions on nearly every new tank in order to reduce warpage on these parts to an acceptable level. Part and mold design modifications, which might reduce the molding problem, may often be discarded because they are a departure from accepted historical methods. The short-term cost, time, and risk of developing new approaches often distract from the potential long-term gains of pursuing them.

The oversized flanges (see Fig. 10.6), typical of these parts, create many of the actual molding and warp problems, as well as create analysis problems for today's injection molding simulation programs. The thin shell–type mesh used by these programs does not represent the low width-to-thickness aspect ratio of the flange very well. Even though the programs are used regularly in the design of new molds for automotive end tanks, these modeling limitations inhibit their full use and require skilled consideration by the analyst.

For structural applications a designer will make use of either standard design formulas for calculating deflections and stress, or, for more demanding designs, use finite element structural analysis. Both of these methods, however, are significantly challenged due to the complex characteristics of plastic materials. Even if a structural analysis program has the sophistication to solve the nonlinear problems required, it is doubtful that the material data will exist for the given material that is to be analyzed.

In addition, there is the unique requirement of the plastic part designer to consider the mechanical properties of the material as well as its melt characteristics. The materials melt characteristics will reflect on the manufacturability of the part. Again, the materials properties during mold filling are expensive and difficult to capture. The viscosity, like the modulus, is nonlinear and will continually change during mold filling. The flow channel is constantly changing as plastic freezes with contact to the relatively cold channel walls, restricting the channel.

It is commonly desirable to prototype a part before committing to production. Again, for an injection molded plastic part, this is not a simple procedure. Although parts may be machined or built by many different methods to produce a prototype, these machined parts will only provide limited information. They will not represent the actual mechanical properties or the problems with acquiring the desired size and shape once they are injection molded. They will not predict the mold filling problems that might be expected during injection molding.

Prototype molds might be built. Again the information gained from this is limited. The molding process and resultant plastic parts characteristics are affected by differences that result from molds built of alternate materials and have different runners, gating, cooling, and processes.

10.2.3 Mold Design and Machining

Molds for the production of plastic parts must be custom designed and built. The construction of these custom-designed molds requires highly skilled craftsmen to operate a wide range of precision machining equipment. The injection mold must replicate the geometry and finish requirements of the part. One of the primary challenges in

the design of a mold is to assure that when it opens, all features of the part are relieved so that it can be ejected. One mold designer told me that 70% of the parts he receives could not be ejected from a mold as designed. This is somewhat of an interpretive point in that some mold designers will tell you they can design a mold to eject nearly any part as long as cost of the tooling and productivity are not a concern. Ejection may require combinations of crafted parting lines and a multitude of movable components that result in a tool that is essentially a fairly complex machine.

The challenge of designing a mold goes much further, however, than does ejecting the part. The mold components must be machinable, cavity dimensions must be sized to account for the parts shrinkage, the mold must provide adequate and uniform cooling, venting of gases, product finish, built to tenths of a thousands of an inch, include delicate inserts, accommodate delivery of the melt, automatic separation of runner and part, and be built like a tank in order to withstand millions of cyclic internal loads from injection pressures in excess of 30,000 psi and external clamp pressures that can reach more than 7000 tons.

The most fundamental requirements are that the cavity can be filled with the specified plastic, be robust enough to accommodate the internal and external forces, and be built so the molded part can be ejected from the mold. The latter two are mechanical requirements that can be more clearly accommodated. Formal structural analysis is rarely performed on the mold designs; rather, the designer relies on skill, intuition, and a conservative approach to structural requirements. Filling of the cavity is less predictable and its effect on the filling and the part are much more abstract. This again results in conservative approaches to the design of the melt delivery system and positioning of gates. This commonly equates to oversized runners, and poor positioning of gates. This conservative approach to addressing the fundamental requirement of mold filling is often in contrast to productivity and product quality issues.

The mold design illustrated in Fig. 10.7 is used to produce a cap that has both an inside and outside thread. Due to this thread arrangement the mold includes an internal unscrewing mechanism, side action, and a stripper mechanism. In this case a simple-looking plastic part weighing approximately 10 g requires a mold weighing in at approximately 300 kg. Despite the eloquence of this mold design and the applaudable engineering that went into it, this mold is limited to producing only two parts per cycle and will result in a weld and gas trap that must be vented through some stationary pin or plug that requires regular maintenance.

10.2.4   Process

After the material has been selected, the part designed, and the mold built, an attempt to produce a part can be made. At this time, one can determine what the product will look like, how it will perform, and how inexpensively it can be produced. This is often the beginning of an iterative cycle, which may include alterations in material, part design, and the mold before again trying to mold a part to expectations.
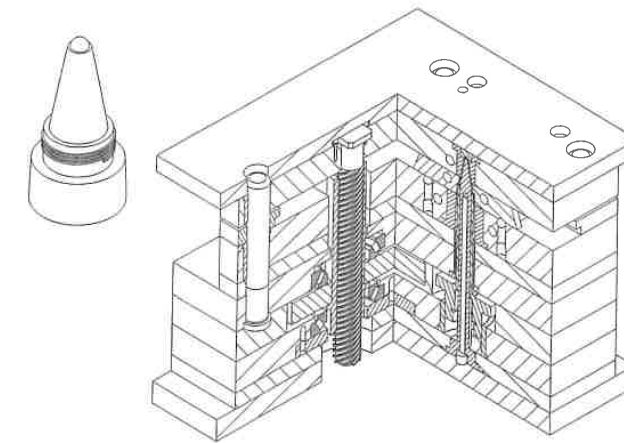
Figure 10.7 Mold including an internal unscrewing mechanism, side action, and a stripper mechanism used to produce a cap with both an internal and external thread (Drawing by Tom Garman).

The size, shape, and mechanical properties of an injection molded plastic part are significantly affected by the molding process. The complex shrinkage of plastic materials is probably the most dominate underlying problem associated with successful design and manufacture. During the injection molding process, plastic materials go through a volumetric change of as much as 35%, as they are cooled from a molten to a solid state. If this did not present enough of a design and molding problem, flow of plastic through the mold induces molecular orientation that introduces a directional component to this shrinkage. The direction and magnitude of the shrinkage varies significantly from part to part, and even within a given part. The direction and magnitude of the resultant shrinkages affects the final size, shape, and mechanical properties of the finished product. It becomes imperative to understand these shrinkage phenomena in order to develop design and molding strategies that will minimize their potential negative effect.

As discussed previously, thermoplastics are comprised of long, organic molecular chains that are made up of numerous repeating carbon based units (monomers). The atoms within these chains are held together by relatively strong covalent bonds. The individual polymer chains are held together by relatively weak electrostatic forces. These weaker secondary forces have only a fraction the strength of the primary bonds that hold together the individual atoms within the polymer chain.

During the injection molding process, a polymer mass is heated to a point at which it melts. In this molten phase, the material can flow and is forced into a cold mold where it will take the shape of the cavity. Following cooling, the material can be removed from the mold, and it will retain the general shape of the cavity.

Shrinkage of thermoplastics can be characterized into two broad classifications, volumetric and linearized. Volumetric shrinkage can be attributed to thermal contraction that will occur in all polymers and crystallization that occurs in semicrystalline polymers. With the absence of any external forces, this shrinkage will be isotropic. Linearized shrinkage is developed from shear and extensional forces acting on a polymer during mold filling and packing. These forces develop orientation in both the polymer and the fillers, which may be contained in them. This orientation can vary in direction and magnitude and has a direct effect on how the polymer shrinks. The following presents the phenomena of volumetric shrinkage resulting from thermal contraction and crystallization and linearized shrinkage as developed by flow in the mold.

Volumetric Shrinkage

Thermal contraction and expansion is a well-known and understood phenomenon, which occurs with most all known matter. With few exceptions, a material will expand when heated and contract when cooled.

When relatively low levels of heat are introduced to a polymer, this external energy increases molecular motion and weakens the cohesive bond energies of both the primary atomic and secondary molecular bonds. The result is an increase in the specific volume of the polymer mass as the atoms and molecules move apart. The amount of expansion will be proportional to the increase in heat input to the body. As the secondary bonds are weaker, the introduction of heat will have a more dramatic effect on them. During subsequent cooling, if no external forces are applied, the polymer would experience uniform orthotropic contraction (shrinkage).

With continued heating, the secondary bonds will continue to weaken until motion between polymer chains become relatively free and the polymer begins to exhibit distinct fluid like characteristics. Through examination of a P-V-T graph, the relationship between pressure, specific volume, and temperature can be evaluated. Figure 10.8 illustrates a characteristic P-V-T relationship of an amorphous material. By following the shrinkage of the material at 0 MPa, one can see that the material shrinks more than 10% as it is cooled from its process temperature of 250°C to room temperature, 23°C. The inflection in the curve indicates the glass transition temperature for this material.

As there is a significant amount of free space as a plastic is heated, applied pressure will result in compression of the material. The effect of this compression at various temperatures can also be determined by examination of Fig. 10.8, which shows the PVT characteristic of a polystyrene. At the process temperature of 250°C, the material can be seen to compress from a specific volume of approximately 1.06 cm³/g at atmospheric pressure to 0.95 at 160 MPa. These pressures correspond to the potential range experienced during the injection molding cycle on a typical injection molding machine. The combination of thermal contraction and pressure yields potential variances of 14% in volume during molding in this particular case.
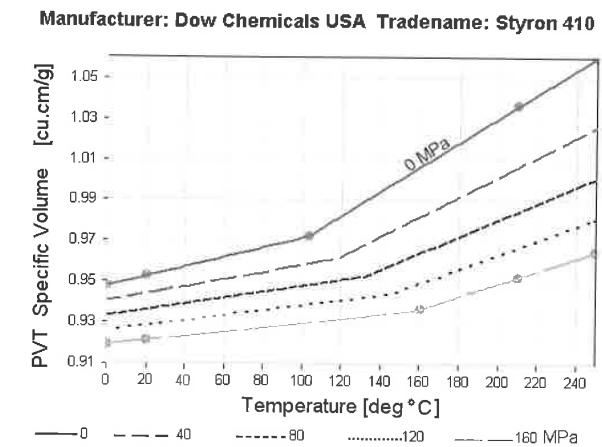
Figure 10.8 Characteristic PVT relationship of an amorphous material.

Volumetric Shrinkage of Semicrystalline Polymers

Ordered polymers, such as HDPE, can form crystals in which the polymer chains fold back upon themselves resulting in densely packed parallel chains. The effect of this can be seen in Fig. 10.9, which is a characteristic P-V-T graph of a semicrystalline polymer. At the elevated temperatures where the polymer is molten, its structure is amorphous. As the material is cooled, initially there is a gradual linear reduction in specific volume. At the materials crystallization temperature, there is a sudden
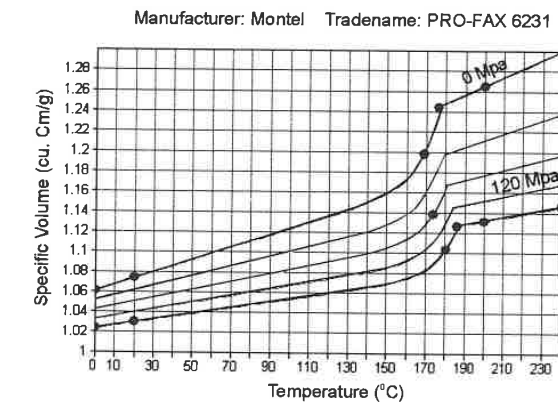

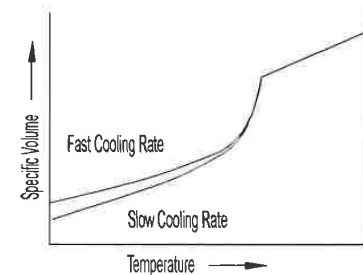Figure 10.9 Characteristic PVT relationship of a semicrystalline material.

Figure 10.10 Illustration of the relationship of cooling rate and shrinkage of a semicrystalline material at atmospheric pressure.

decrease in specific volume as the polymer transforms from a molten amorphous to a solid semicrystalline structure. Once the material has solidified, the reduction in specific volume with change in temperature again becomes more gradual and linear. In this case, the volumetric change is nearly double the amorphous material under similar conditions.

The formation of the crystallization structure requires that the molecules must organize themselves from the random state that exists while molten. As this physical structuring of molecules requires freedom of movement during the time that the crystals are forming, the rate at which the polymer is cooled will effect how much of this structuring occurs (i.e., what percentage of the polymer will be crystalline versus amorphous). It can also affect the size of the forming crystals. As the crystals are denser than the amorphous regions of the polymer, it can be expected that the rate of cooling will effect how much the polymer will shrink. Therefore, unlike amorphous materials, the density of a semicrystalline material will be affected by cooling rate. Figure 10.10 illustrates the relationship of cooling rate and shrinkage of a semicrystalline material at atmospheric pressure. The differences in specific volume at room temperature result from changing the cooling rate of the molten semicrystalline polymer. Also note the abrupt change that occurs in specific volume between its melt temperature and room temperature. This is the region in which the polymers crystals are forming during cooling (disassociating during heating).

Linear/Directional Shrinkage

During the injection molding process the polymer is subjected to thermal energy and plasticated in the injection barrel. The molten plastic is then forced under high pressure into a cold mold. The resultant shear field, acting on the expanded polymer mass, results in the molecules becoming oriented in the direction of the principle strain. The degree of orientation is a function of the applied shear stresses that are commonly over 100,000 pascals. This orientation, or ordering of the molecules, results in a relatively high energy state which in effect reduces its entropy. Given time, the molten mass would lose its orientation and return to a random state. This restoring force may be described as entropy elasticity. As the dominant orientation is normally

in the direction of flow, this returning to a random state, or elastic recovery, would effectively reduce the length of the mass in the direction of the original applied stress (i.e., increase the shrinkage in the direction of flow versus transverse to flow).

In response to economic goals, a mold is kept relatively cold to minimize the time it takes to form a part, and thereby its apparent cost. This results in the outermost layers of the laminar flow front freezing nearly instantly as they contact the cold steel wall of the cavity. This freezing action takes place so quickly that the rapid reduction in free volume restricts the expected effects of entropy elasticity. Immediately under this frozen layer, the material is insulated from the steel wall and therefore cools more slowly. This allows time for recovery to take place, which effectively results in the expected linear or directional shrinkages resulting from flow. These variations in linear shrinkages are continuous through the entire cross-section of any given region, or element, of the part and are set up during the filling, packing, and compensation phases of the molding cycle. As these laminates with variable shrinkages are mechanically linked, a stress will develop between them.

From the above, it is apparent that if unconstrained by adjacent laminates within any given cross-section of a molded plastic part, the material would shrink differently within each discrete laminate. The orientation in the outermost laminates will be higher as a result of the stronger shear field. Figure 10.11 is the results of a flow analysis providing information on Shear Rate distribution through a cross section during filling. These outer laminates are also the first to cool and thereby lock in the orientation.

During the compensation phase of the mold filling cycle, the shear field will move more toward the center as the outer laminates quickly freeze and thereby locking in the high orientation. However despite the increase in the shear field near the center, the rate of cooling is much slower and thereby allowing the material to relax (disorient). The result is that the net shrinkage is a balance of the stain between each of the laminates.

These differences in stains also result in a stress between laminates. Studies have been conducted which have looked at the stress distribution through the cross section of plastic parts [1]. These studies have evaluated stresses by studying the birefringence of a specimen through its cross-section. The results have shown that the outer
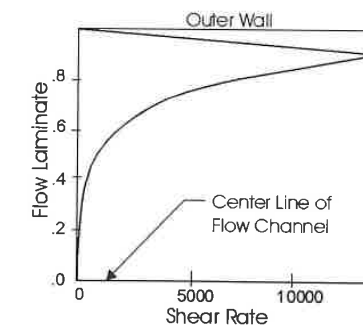

Figure 10.11 Shear rate distribution through a cross-section of the melt flow during filling. Orientation is highest in the outer laminates.
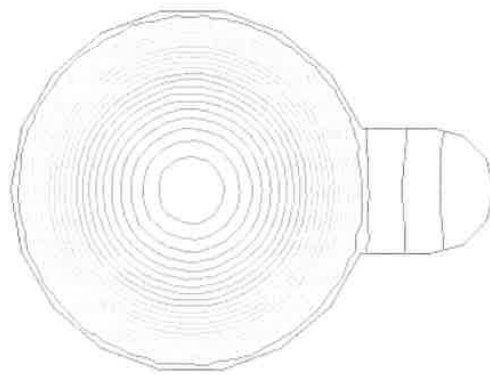
Figure 10.12 Filling pattern during injection molding. Each line represents the melt front advancement at equal increments of time.

laminates are under a compressive stress while the center laminates are under a tensile stress. This results from the conflicting strains between these regions.

Linearized shrinkages resulting from flow are complicated further by changes in flow direction and flow velocity that often occurs through the cross-section during filling and compensation phase.

Figure 10.12 is the filling analysis results from a simple part that is gated in the center of the large circular region. The geometry is used to describe the complex flow conditions that can be established, even in this simple shape, and how this affects shrinkage. Each of the lines is referred to as isochrones and represent the advancement of the melt front at equal increments of time. Close spacing represents slow movement; further spacing represents rapid movement. The part is filled from a single gate at a constant injection rate. As the melt emerges from the gate the velocity of the flow front decreases as it diverges in a radial pattern through the increasing volume of the part. Upon reaching the edges of the large circular main body of the part, the flow becomes limited to filling the appendage on the right side. At this moment all flow entering the cavity is redirected towards this last region to fill. This results is a sudden surge in the flow front velocity into the appendage. This velocity increase between the gate and the appendage causes an increase in shear stress in this region. The high shear stress is shown in Fig. 10.13 as the lighter colored region. Additionally the sudden increase in velocity results in a pressure spike. Figure 10.14 illustrates the resultant pressure spike, which in this case nearly triples the filling pressure during the last 15% of fill time. With the changing velocities and stresses, there is a resultant increase in the degree of orientation taking places in the flow path between the gate and through the appendage. This in turn will result in variation in the magnitude of the directional shrinkage throughout the part. Therefore we have variations not only in the direction of the shrinkage but also in its magnitude.

The shrinkage described above is complicated even further by changes in flow direction that occur under the frozen layer as the part is filled. Figure 10.15 is a plot
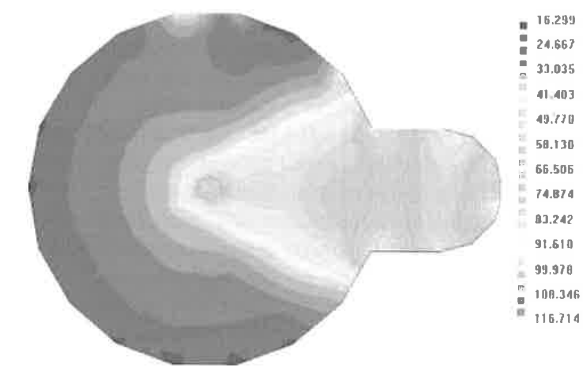
Figure 10.13 Shear stress distribution through the part as a result of melt front velocity.

that provides information on the direction of melt flow at the instant of fill. This is the direction of flow under the frozen plastic layer that was formed during the original mold filling. When contrasted to the original filling pattern, Fig. 10.12, one can interpret where flow direction, and resultant orientation, will be different between the outer laminates and the inner laminates of the molded part. The orientation of the outer laminates will be controlled primarily by the original filling pattern, while the orientation of the inner laminates will be primarily controlled by the later flow conditions under the frozen layer. One can see that in some regions of the part the flow direction remains constant throughout filling while others have changed significantly under the frozen layer. Therefore it can be expected that not only does linear
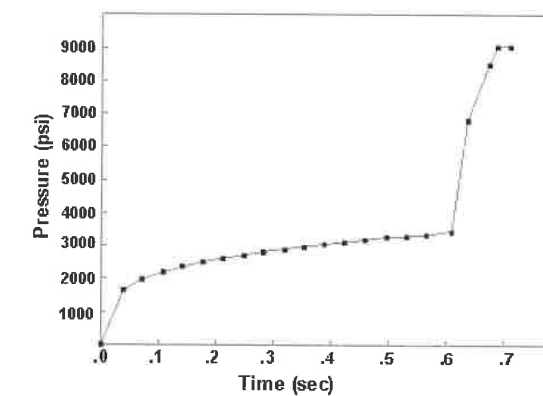

Figure 10.14 Pressure to fill the cavity nearly triples at the same instant the melt front velocity increases.
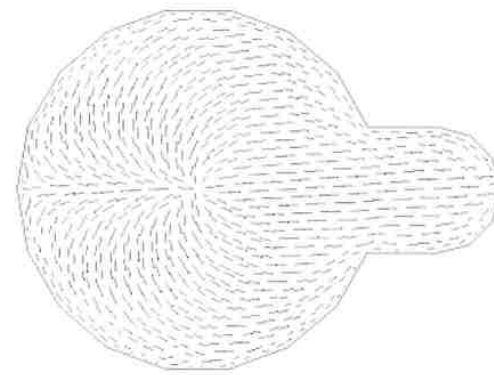
Figure 10.15 Plot providing information on the direction or angle of melt flow at the instant of fill. This is the direction of flow under the frozen plastic layer that was formed during the original mold filling.

shrinkage vary 360 degrees globally throughout this part, but that within any given cross section the shrinkage direction and magnitude can be different.

Shrinkage of Fiber Filled Materials

The inclusion of fibers (typically glass or carbon) can have a significant effect on the linear shrinkage of a polymer during injection molding. The asymmetric ridged nature of the fibers restricts shrinkage of the polymer matrix in the direction of the fibers orientation. Therefore, if the fibers become oriented in the direction of flow, then the material will be restricted in its shrinkage in that direction. The result is the highest shrinkage will take place in the transverse to flow direction rather than the common, high flow direction shrinkage. In cases where there is a high degree of orientation of fibers, the relative shrinkage of flow and transverse to flow can be quite pronounced.

Figure 10.16 is the results of a study by B. G. Jaros [2] on the anisotropic shrinkage developed in parts molded of a 30% glass fiber filled PBT. Shrinkage in three different geometries were contrasted. The three geometries created variations in flow type. The first geometry created a near ideal linear flow. The test specimen was 50-mm wide, 200-mm long, and 2-mm thick. The part was fed from one end by a specially designed fan gate so that the flow would be linear across the entire length of the part. Under these conditions the resultant shear field will act on the glass fibers, as with the polymer chains, and orient themselves along the length of the sample. Note that the average shrinkage, labeled LINEAR, in the flow direction is 0.18% whereas the transverse to flow shrinkage is 1.7%. This is a variation of more than 9:1 (transverse to flow:flow).

Glass fiber orientations can be significantly affected not only by the direction of flow, but by whether the flow is dominated by shear or extensional flow effects. In the case above, flow within the cavity is almost exclusively shear dominated. Virtually no extensional flow exists. This can be contrasted to a center gated disk. In the case of the disk, the melt enters into the center and expands outward as it fills the cavity. As the melt diverges through the cavity, its flow is not only radial from the gate, but in fact, has a component which acts perpendicular to this direction which is referred to as extensional flow. The condition is analogous to the surface of an expanding balloon where the surface is stretched perpendicular to the direction of its expansion. This extensional flow will act on the glass fiber and result in orientation in the transverse to flow direction.

The net orientation of the glass fiber will depend on whether the dominant force acting on any given fiber results from shear flow or extensional flow. In the outermost laminates adjacent to the cavity wall, it is expected that shear stresses will be the dominant factor. Midway through the cross-section of the flow channel the shear field is nonexistent. As a result the tensile forces of the expanding flow front will be dominant and orient the fibers perpendicular to flow. The cross over as to which is dominant will depend on process conditions and location within a mold. In Jaros's study, radial flow effects on shrinkage was evaluated using a 101.6-mm radiused half disk. The same 30% glass fiber filled PBT was run in this mold with the same thickness as with his linear flow cavity. This radial flow shrinkage data is summarized in Fig. 10.16. The average flow direction shrinkage was found to be 0.33% and the
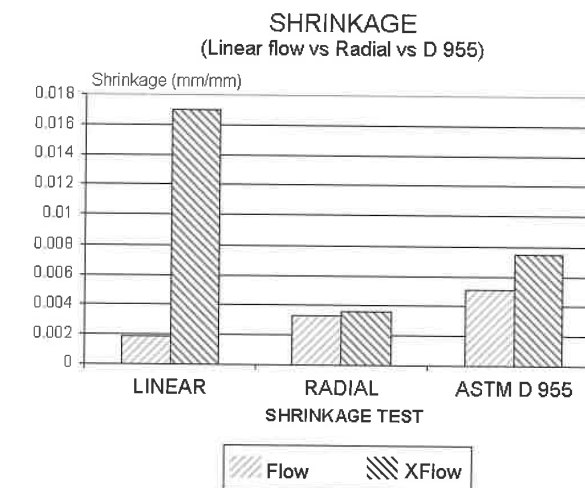

Figure 10.16 Results of anisotropic shrinkage study developed by B. G. Jaros [2] in parts molded of a 30% glass fiber filled PBT.

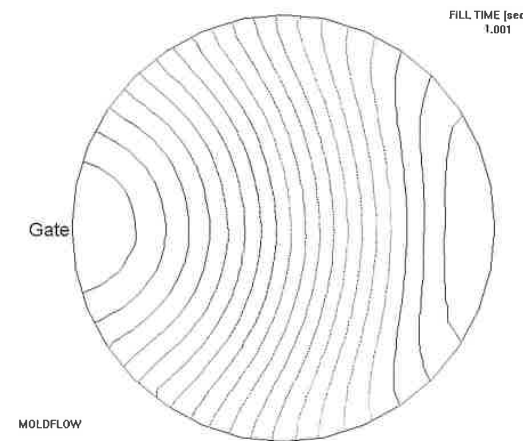FILL TIME [sec]
1.001

Gate

MOLDFLOW

Figure 10.17 Filling pattern of 102 mm diameter test plaque. Initial melt front is radial as it diverges from the gate.

average transverse to flow was 0.36%. This is a variation of only 1.09:1 (transverse to flow:flow) versus the 9:1 ratio found in the pure linear flow part.

Finally the shrinkage results from the linear and radial geometries that Jaros molded are contrasted to the data provided for the same material by the material supplier. The material supplier's flow and cross flow shrinkage data was generated from a standard 102 mm diameter test plaque. The sample is edge gated from a narrow tab gate.

As the flow enters the standard test plaque cavity, its flow is initially radial as it diverges from the gate (see Fig. 10.17). This initial extensional flow establishes cross-flow orientation of glass fibers in the midstream of the cavity. As the flow front progresses, its direction begins to become linear through the remainder of the part. However, in the absence of an offsetting shear force in the midstream, the cross-flow orientation will be maintained through filling of the rest of the part. This results in complex variations in orientation not only through a given cross-section of the part but globally across the part. The result is that the standard test method provides data showing only a difference of less than 1:1.4 in shrinkages (flow:cross flow). This is in contrast to Jaros's data, which was based on pure linear and pure radial flow data.

Determining Mold Cavity Dimensions

As discussed previously, shrinkage of plastics will vary both in magnitude and direction throughout a molded part. Therefore determining the required cavity dimensions to achieve the specified size of a plastic part can be extremely elusive. As a result, it is common that a cavity has to be modified numerous times to achieve the specified part size. This is an extremely time consuming and costly task which can far outweigh the cost of the initial tool.

Most mold cavities are sized based on data collected through standardized test methods such as ASTM D955. This data must be used with caution, as the source of

this data virtually never represents the geometry or process of your molded part. It is common that only a single shrinkage value may be known for a material. This value is often used regardless of flow length, cavity thicknesses, or process. Based on Jaros' study, published data for a 30% Glass Filled PBT developed using ASTM D955 states that shrinkage ranges from 0.4% to 0.8% depending on flow direction and process. Jaros's test data on the same material found significantly different shrinkages. Jaros's samples were molded using a wide range of processing conditions with two different cavities creating either pure linear or pure radial flow conditions. Depending on flow type and process, Jaros found that shrinkage with the same material varied from 0.15% to 2.04%.

Correctly dimensioning mold cavities is a common problem in the plastics molding industry. Mold shrinkage analysis software is attempting to remedy this situation. However, most mold builders find it difficult to justify the cost of performing a shrinkage simulation. Additional problems include the limited availability of material's characterized for shrinkage, the time to conduct the simulation and the potential for error in the simulation. All of these limitations are continuing to degrade with time and the further development of the technology.

The most common method of sizing cavities today remains using hand calculations based on published shrinkage data provided by the material supplier. The following formula is used when calculating required cavity dimensions from a fixed shrinkage constant.

$$L_M = \frac{L_P}{1 - \alpha_L}$$

Where L_M is the length of the mold cavity, L_P is the target length of the molded plastics part and α_L is the shrinkage coefficient for the given material.

Sample Problem: Given a molded part that is to be 610-mm long. The shrinkage coefficient (α_L) for the given material is 0.021 mm/mm. The required mold cavity dimension is found by:

$$L_M = \frac{L_P}{1 - \alpha_L} = \frac{610 \, mm}{1 - 0.021 \, mm/mm} = 623.08$$

Shrinkage coefficients for some common polymers are shown in Table 10.1. It should be realized that these values can vary dependant on process, geometry, material grade, material supplier and types and amount of additives.

Shrinkage during the injection molding process can also have a positive contribution. An example is in the ejection of parts from the mold. Shrinkage of the part onto a core controls which side the part will adhere as the mold opens. This allows for the design of an ejection system for positive removal.

Warpage in Plastic Parts

Warpage of plastic parts results from variations in shrinkage. These variations create stresses that can overcome the mechanical properties of the molded plastic part and

Table 10.1   Sample Shrinkage Values Used for Neat Injection Molded Thermoplastics

Name of material	Shrinkage (mm/mm)
ABS	0.004–0.006
Reinforced ABS	0.001–0.003
ABS/PVC alloy	0.004–0.006
Acetal	0.020
Acrylic	0.002–0.006
Modified acrylic	0.002–0.006
Cellulose acetate	0.005–0.008
Cellulose acetate butyrate	0.003–0.006
Teflon TFE	0.030–0.070
Teflon FEP	0.040–0.060
Nylon Type 6	0.010–0.015
Nylon Type 66	0.015–0.020
Reinforced nylon	0.002–0.005
Phenylene oxide-based resin	0.005–0.007
Polycarbonate	0.005–0.007
Reinforced polycarbonate	0.003–0.005
Polyaryl ether	0.003–0.007
Polyesters (PTB)	0.004–0.008
Reinforced polyesters	0.0025–0.0045
Polyether sulfone	0.0015–0.003
Polyallomer	0.001–0.002
Polyethylene Type I	0.030
Polyethylene Type II	0.030
Polyethylene Type III	0.015–0.040
Polyethylene Type IV	0.015–0.040
Reinforced polyethylene Type IV	0.003–0.005
EVA copolymer	0.010
Polyphenylene sulfide	0.001–0.004
Polypropylene	0.010–0.020
Reinforced polypropylene	0.003–0.005
Polypropylene copolymers	0.010–0.020
Polystyrene general purpose	0.004–0.006
Polystyrene impact	0.004–0.006
SAN copolymers	0.003–0.007
Reinforced SAN copolymer	0.001–0.003
Polysulfone	0.007
Reinforced polysulfone	0.001–0.003
PVC rigid	0.004–0.006

cause it to distort. The ability of the plastic part to resist these residual stresses depends on the rigidity of the plastic material at the time of ejection and the rigidity of the parts shape. If a part is ejected from the mold while it is still relatively hot, the modulus of the material will be lower and therefore more prone to warping. Holding the part in the mold longer will lower the temperature of the plastic and increase the modulus of the plastic material. The shape of the plastic part can also have a significant effect on whether the part will warp. A flat part has little structure and is much more susceptible to warpage than a part like a cup which has good structure. Ribs are sometimes added to a flat part to add rigidity and thereby resist warpage. However, the ribs themselves can often create variations in shrinkage and have an opposite effect, thereby inducing warpage.

Mold and molding factors that affect warpage by creating variations in shrinkage include:

- Direction and magnitude or orientation. These factors are affected by the following.
  - Fill pattern indicates direction of orientation on the outer laminates of a molded part.
  - Transient flow (effects the direction of orientation in the midplane of a molded part.
  - High shear stress results in high orientation. Regions of high shear stress are commonly near gates and the end of fill where melt may accelerate into restricted regions.
- Uniformity of cooling. Variations in cooling can occur from side to side or region to region within a plastic part. Side to side variations will create a bending conditions with the part warping toward the hot side. Regional variations in cooling can create an unstable buckling condition.
- Pressure variations across a part. It is normal that pressures will be highest near the gate. Given a part with a constant wall thickness, it can be expected that volumetric shrinkage near the gate will be less than regions away from the gate. In a center gate circular part, like a disk, this would tend to cause the part to warp in a bowl shape.

Residual Stresses in Plastic Parts

Variations in shrinkage in a part will create residual stresses. As indicated above, if these stresses can overcome the rigidity of the part, it will warp. However, regardless if the part warps or not, stresses will remain in the part. Over time, these stresses may cause premature failure through a multitude of factors. The same mold and molding factors identified as contributors to causing warpage will create residual stresses. The residual stresses developed from variations in shrinkage can add to the stresses developed from external loading. Additionally, environmental factors such as chemicals or temperature can accelerate the effects of residual stresses, causing unexpected premature part failure.

Mechanical Properties

The mechanical properties of a molded part can be effected by both flow induced orientation and the existence of weld lines created when two melt fronts combine. Orientation creates anisotropic mechanical properties which are much like the effect of the grain structure in a piece of wood. Under flexural loading, a part will be stronger when loaded perpendicular to orientation. Under tensile loading, the part will be stronger when load parallel to orientation. Like orientation induced anisotropic shrinkage, these variations in mechanical properties are most severe with fiber reinforced materials.

Weld lines are formed when two melt fronts join. As there is at least a reduction in molecular orientation across a weld line, there will generally be a reduction in mechanical strength along the junction. A butt weld, where the flow meets straight on, will be weaker than a meld line, where the two flow fronts meet at an angle and merge together. A fiber reinforced material will have a significant reduction at a weld. Generally the higher the fiber content the higher the percentage loss in strength. A transient flow can develop across a weld and effect its strength. This is a condition in which a flow continues through the weld, under the outer frozen layers, after the melt fronts initially meet. The continued flow causes molecular and fiber orientation across the weld and can have dramatic effects in increasing weld strength. This can be induced by special tooling or processes. In some cases this transient flow can be created by strategic gating within the mold.

10.3   Part Design Guidelines for Injection Molded Plastic Parts

General guidelines for designing plastic parts have evolved over the years and are focused at issues related to manufacturability. These can include consideration of material shrinkage, part ejection, cooling, and mold filling; however, one will find that many of these guidelines contradict each other. A guideline developed to address one problem can create a new, and sometimes more severe, problem. It is important, therefore, to understand the logic behind the guidelines and the potential consequence of utilizing them. With this understanding a designer can make informed decisions regarding a design and anticipate the consequences of including a specific feature in the design. This may cause the designer to seek an alternative, improved approach to achieve the products objectives.

The most troublesome issue related to successfully developing a new plastic part is probably anticipating how it will shrink and warp after molding. Many part design guidelines attempt to address these issues, but fall far short. Shrinkage of plastic parts will basically vary from material to material and within a given material. This makes it difficult to design a mold and process that will produce a part to the desired size. In addition, variations in shrinkage within a given part develops residual stresses that act to warp the part. The stresses that do not warp the part will reside in the part and potentially cause delayed dimensional and structural problems. The more the shrinkage variations within a part, the more potential there is for warpage and residual stresses.

High-shrink, and fiber-filled, materials create more potential for shrinkage related problems than do low-shrink, and nonfiber-filled, materials. A high-shrink material has a higher potential for shrinkage variation than does a low-shrink material. Unfilled, or neat, amorphous materials are relatively low-shrink materials, with most of them having linear shrinkages of less than 0.010 mm/mm. Neat semi-crystalline materials will normally have high shrink values ranging from 0.010 mm/mm to more than 0.050 mm/mm. These shrinkage values will vary for each material. They will also vary for each material dependent on part design, mold design, and process.

Polymer additives, such as rigid fillers and reinforcements, will reduce overall shrinkage. If the additive has no real aspect ratio, it will generally reduce shrinkage-related problems. Fillers with an aspect ratio, such as a fiber where the length is longer than its diameter, will reduce overall shrinkage but create significant variations in directional shrinkages. These directional shrinkages develop from orientation established during mold filling and compensation phases. The result is that fiber-filled materials will tend to increase a materials desire to warp.

One of the primary objectives for the designer of injection molded plastic parts is to maintain a uniform wall thickness in the part. All designs should be approached with this objective in mind. A uniform wall will minimize injection molding problems, particularly those related to shrinkage. It is common that a plastic part be designed such that it includes relatively thick features attached to thinner wall sections. This is a major source of many part design problems. A designer should make every effort to eliminate or core out any thick features in a plastic part.

Most of the following design guidelines are based on how to combine various design features of a part with the primary wall. This will include how to add features such as ribs, bosses, and holes. The addition of ribs that are classically thinner than the primary wall is of particular interest, as they break the most fundamental design objective of maintaining a uniform wall thickness.

Before applying any design feature, or guideline, that creates variations in a parts wall thickness, the designer should ask:

1. What is the reason for modifying the thickness of this feature relative to the primary wall?
2. Does the reason for the variation have significance in this part?
3. Is there an alternate approach that will avoid this variation?

4. If the feature creating a variation in wall thickness must be used, what are the potential problems created by this variation and what can be done to address these problems?

After answering these questions, the designer must judge the benefits, or needs, of varying the wall thickness against the potential problems that will be created. This chapter is intended to help a designer make these judgments. It should be understood, however, that *there are no fixed rules in designing plastic parts*. The following guidelines are suggestions to minimize problems, but they must always be judged individually for a particular case.

10.3.1 Designing the Primary Wall

Injection molded plastic parts are generally designed around the use of relatively thin walls. Unless the injection molded plastic part is foamed, or is produced with gas assisted injection molding, its walls are normally less than 5 mm (0.2 in). When determining the thickness of the primary wall one must consider the structural, functional, and aesthetic issues related to the wall as well as its impact on manufacturability. Manufacturability issues include consideration of the injection pressure required to fill the mold cavity, cooling time, and the influence on ejection from the mold. These will require consideration of available injection pressure and injection rates, mold rigidity, ejection techniques, and melt delivery means.

Maintain a Constant Wall Thickness

Maintaining a constant wall thickness should be the primary objective of the product designer. Each region in a part that has a different thickness will want to shrink differently. These variations in shrinkage will complicate achieving the desired size of the molded part and be major contributors to warpage and residual stresses. Variations in wall thickness also affect the mold filling and packing, or compensation, phases of the molding cycle. Irregular, and difficult to predict, filling patterns can result. This can further complicate orientation-induced shrinkage as well as cause problems with venting, gas traps, and weld lines. In addition, a thick region fed through a relatively thin region can result in sinks and voids in the thicker region. A thin region, such as a rib, attached to a thicker region can result in hesitation effects during mold filling that could potentially cause no fills.

If variations in wall thickness cannot be avoided, then try to keep the variation to a minimum and provide a gradual transition rather than a sudden change (see Fig. 10.18). Be conscious of the negative effects of having these variations that would include variations in shrinkage, warpage, residual stresses, and undesirable mold filling patterns.
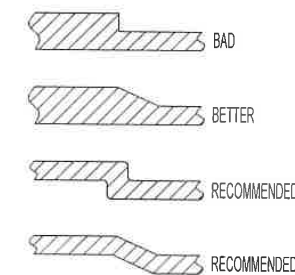
Figure 10.18 Provide as gradual a change as possible (or eliminate the change if possible) at the intersection of a thick/thin region.
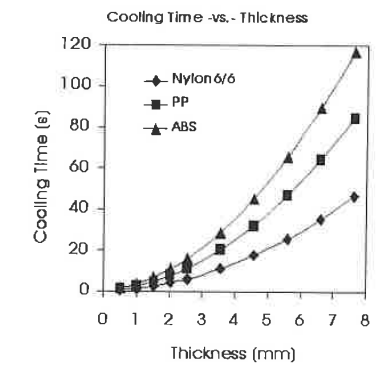

Figure 10.19 Exponential increase of cooling time with respect to wall thickness for three common plastic materials.

Minimize Wall Thickness

For injection molding, thinner walls will lead to more cost-effective production through the use of less material and shorter molding cycles. The plastic material used and the cooling time can often be attributed to more than 70% of the part's cost. Reducing the amount of plastic material has an obvious direct proportional effect on reducing material costs. Reducing cooling time through a thinner wall has an even greater potential benefit, particularly where cooling time is a significant portion of the part cost. It can be seen from Fig. 10.19 that cooling time increases exponentially with an increase in wall thickness. Doubling wall thickness of an ABS part from 1.0 to 2.0 mm increases cooling time by approximately 2.6 times. Doubling the wall thickness again, from 2.0 to 4.0 mm, increases cooling time approximately 3.8 times.

The overall part thickness is generally determined by the amount and type of loading that the part is expected to withstand. Depending on the application of the part, other factors that should be considered include electrical, sound and thermal insulation, and gas permeability. Regardless of the particular requirements, it should be the objective of the designer to minimize the wall thickness while satisfying the part's functional requirements. The designer must be conscious of the relationship between wall thickness and pressure to fill in accomplishing this. Reducing wall thickness will exponentially increase pressure to fill. Halving the wall thickness of a part can easily cause a threefold increase in pressure to fill.

In applications where plastic parts are to replace metal parts it is common that the plastic part will be cored out relative to the original metal part its replacing. Metal parts are often made bulky and solid, as their formation normally requires machining, or removal of material from a solid chunk of steel. This is a subtractive process where increasing the amount of steel to be removed increases machining time and related costs. The more steel removed, therefore, the more expensive the product. In
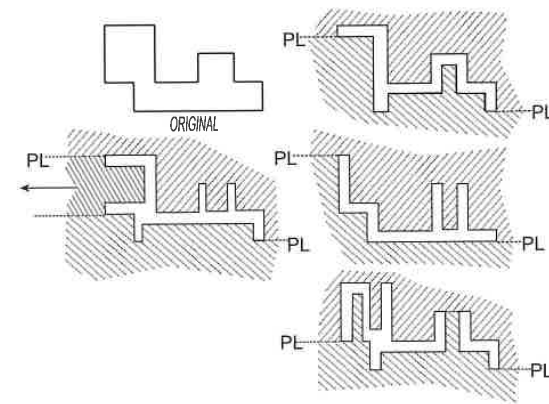
Figure 10.20 Design alternatives are shown for a part with thick regions. The optimum option will depend on the functional and structural requirements of the part. The molds parting line and ejection requirements must be considered.

contrast, injection molding of plastics parts is an additive process. Here, however, the opposite is true: the less the material, the lower the cost of the product.

Figure 10.20 illustrates four alternative designs for a part that might have been designed to be machined from steel. The three design cross-sections on the right would be easily produced with injection molding and are shown as cavities formed between two mold halves that would open vertically. Consider how the need to remove the molded part easily has influenced the design. The alternative design on the left bottom could also be easily produced, but it would require a side core. Without the side movement of the side core, the feature on the left would create an undercut that would trap the part in the mold.

Shape of the Wall

The shape of the wall can affect the rigidity of the part and the potential for warp. A domed or circular shape is much more structurally stable than a flat part. A flat part has little structure with which to resist warpage. When molding a flat part, therefore, one must be more concerned with establishing a flow pattern, during mold filling, and a process that will minimize stresses. It should also be expected that successfully molding parts, with shapes that are prone to warping, will be more sensitive to changes in process and are likely to result in higher spoilage.

The shape of the wall can also affect cooling time. A nonstructural shape will be more sensitive to warping. To resist warping, the part must be constrained in the mold longer while it cools further and becomes more rigid. The cooler, more rigid part will be able to better resist distortion from its residual stresses. Regions in a part formed by tall cores will increase the local thermal load, making cooling more difficult.

Features such as flanges, positioned along the perimeter of the part, can increase rigidity without adding thickness. These flanges should ideally be perpendicular to the primary wall to which they are attached, and be of the same thickness or slightly thinner. A thicker flange around the perimeter of a part, like a disk, will cause the center to buckle, forming a bowllike shape.

Surface Finish

Polishing the core in the direction of draw (ejection) is normally recommended to assist in ejection when small draft angles are required. The exception is polyolefins and elastomers where a courser 400 grit polish is recommended.

10.3.2 Ribs, Gussets, and Bosses

Ribs, gussets, and bosses are features that are added to the primary wall for structural, assembly, or other functional reasons. When adding these features to the primary wall, the plastics designer must again consider manufacturability. This includes consideration of part ejection, venting of air during mold filling, and effects on mold filling and packing. The driving force behind most of the standard design guidelines for these features is the attempt to balance these manufacturability issues that are often conflicting.

Ribs

Ribs are primarily used to increase the rigidity of a part or a specific region of the part. To achieve the desired rigidity of a given part an excessive thickness might be needed in the primary wall. This would negatively affect part cost. The addition of ribs can often attain the required rigidity while maintaining the more desirable thinner wall.

From a manufacturing standpoint the ribs should be kept as short as possible and be drafted (i.e., tapered on either side) (see Fig. 10.21). Both of these requirements ease ejection problems. In addition, a short rib reduces mold filling problems and excessive thinning of the rib tip, resulting from the required draft. Several short ribs, therefore, are preferred to one tall rib.

The intersection of the rib to the primary wall creates an increased volume (region "A" in Fig. 10.21). This is compounded by the need to radius this intersection to minimize stresses. The increased volume at this intersection is generally filled by material flowing through the thinner primary wall (region "B" in Fig. 10.21). During the compensation, or packing, phase of the molding process, this thinner wall will freeze off first, thereby blocking compensating flow to the still shrinking region "A." This will cause sinks and/or voids to form at the intersection. In addition, the thicker region can act as an undesirable flow leader. To address these problems, ribs are normally thinned relative to the primary wall.

Figure 10.22 illustrates a recommended rib design. The base thickness of the rib should be 50 to 75% of the primary wall thickness, depending on a material's shrinkage characteristics. A rib thickness of 50% of the primary wall is generally recommended for high-shrink materials, and 75% for low-shrink materials. High-shrink materials are normally considered those with shrinkages of more than 0.015 mm/mm. Low-shrink materials are those with shrinkages less than 0.010 mm/mm. Because shrinkages can vary from less than 0.001 mm/mm to more than 0.05 mm/mm, the
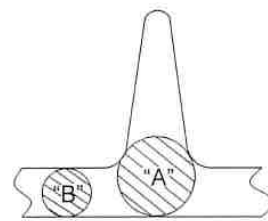
Figure 10.21 The required draft and radius used for a rib results in the large volume of material at region "A" relative to the adjoining primary wall "B." This will create packing problems for region "B."

designer must make judgments as to how to apply these guidelines. There is no hard rule that will work in all cases.

The base of the rib should have a radius. The size of the radius should be 0.25 times the primary wall thickness with a minimum radius of 0.25 mm. The height of the rib should be no more than 2.5 to 5 times the primary wall thickness. Draft angles range from 1/2 degree to a maximum of 2 degrees, with 1/2 to 1 degree used most commonly. Excessive rib thickness and radii should be avoided because these may lead to sinks (or voids) and extended cycle times.

Figure 10.23 illustrates the potential problem created by the improper combinations of draft angle, height, and rib base thickness. The thin top edge can result in both a structural and a mold filling problem. Figure 10.24 illustrates the stress distribution through a rib under flexural loading. It can be seen that the greatest stress is at the top, or free, edge of the rib. Because this region experiences the highest stresses under load it is important that it is not thinned too much.

Design Example:

Given a rib to be attached to a 2.5-mm thick wall, molded of a high-shrink material. The rib should be dimensioned as follows:

- Thickness at base = 1.25 mm
- Height (Maximum) = 6.25 mm
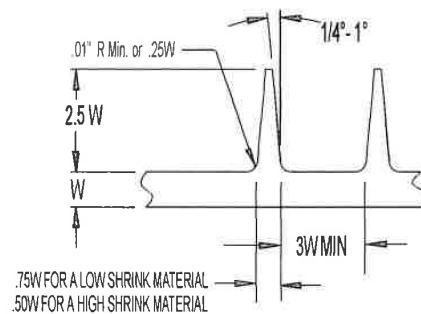
Figure 10.22 Guidelines for designing a rib with respect to the primary wall thickness of the part.
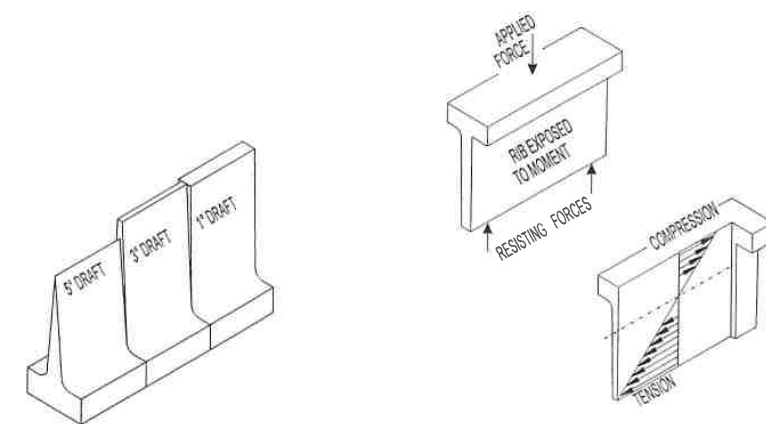
Figure 10.23 Excessive draft on a rib will restrict heights of ribs and result in excessively thin, fragile, free edges.

Figure 10.24 Illustration of the stress distribution through a rib under flexural loading. The greatest stress is on the thin, free edge of the rib.

- Radius at base   = 0.625 mm
- Draft   = 0.5°/side

This results in the top, free edge, of the rib being 1.141-mm thick.

Gussets

Gussets are thin features, much like a rib, that are normally used to reinforce a local feature in a part. The reinforced feature could be a side wall, boss, or some other projection. Gussets are normally triangular in shape and should be designed using the same guidelines for thickness and radii as they are for a rib. In addition, they should be no more than four times the primary wall thickness in height and two times the primary wall thickness in width. Again, this is to address ejection and mold filling problems. Standard design guidelines for gussets are shown in Fig. 10.25. Here, two gussets are shown supporting a boss that is also connected to the primary wall with a single rib.

Design Caution

A major flaw in the previously presented design guidelines for ribs and gussets is that they create a variation in wall thickness that will often cause a part to warp. Ribs, in particular, can often span the entire length or width of a part. The thicker primary wall will take longer to cool and will shrink more than will the thinner rib. As a result, the part will tend to warp away from the rib. Ribs added to flat plastic parts to add rigidity to minimize warpage can actually have the opposite effect and be the cause of warpage.

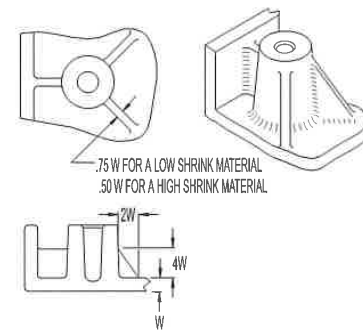.75 W FOR A LOW SHRINK MATERIAL
.50 W FOR A HIGH SHRINK MATERIAL

Figure 10.25 Standard guidelines for gusset design.

As a result a designer must be sensitive to application and aesthetic issues related to the part. If cosmetics are not a concern, reducing the rib thickness becomes less important. A resulting sink from a thicker rib will have virtually no negative structural consequences. This is particularly true when contrasted to the stresses that would develop between a primary wall and a thinner rib. Voids also may be tolerated in many parts at the intersection of a rib to the primary wall. During flexural loading the highest stresses are on the tip of the rib. Next would be the outermost surface of the primary wall opposite the rib. It is most likely that the part failure would be at the tip of the rib rather than due to a sink located in a relatively low stress location.

A further consideration should be the rigidity of the material. A sink is formed when the frozen material on the surface of the part is drawn in as the material below shrinks. A low-modulus material will easily be drawn in by the shrinking material. A high-modulus material that has been cooled will resist having the outer skin being pulled in. As a result, the high-modulus material is more likely to form a void and the visibility of a sink might be minimized. Design guidelines for heights of ribs and gussets are broken all the time with parts still being successfully molded. A designer, however, should be cognizant of the potential problems created when deviating from the guidelines and be prepared to address them.

10.3.3 Bosses

Bosses are normally either solid or hollow round features that project off the primary wall. They can be used for assembly with self-tapping screws, expansion inserts, force-fit plugs, drive pins, positioning, and the like. Bosses are often supported with ribs, adjoining walls, or gussets. Bosses can be free standing or connected to the side walls using ribs. A boss should not be attached directly to a side wall (see Fig. 10.26)
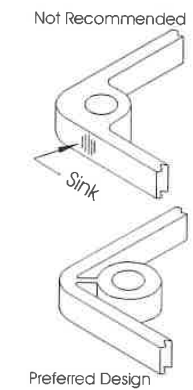
Figure 10.26 Comparison of a boss attached directly to the side wall of a part (not recommended) and the preferred design in which the boss is offset from the wall.

because the intersection with the wall will result in a thick region, causing sinks or voids. A hollow boss can result in weld lines on the outside show surface of the part because the material flows around the core, forming the hollow center. In addition, the resultant weld will commonly form a butt weld. This type of weld is particularly prone to structural failure if a hoop stress is developed from the insertion of a screw or stud used for assembly. The boss can be placed near the side wall and attached to it with a rib that will provide both rigidity and a means for venting (air can travel from the boss, across the rib, to the side wall, where venting is generally available at the natural parting line of the mold).

A standard recommended boss design is shown in Fig. 10.27. The inner diameter of a boss is normally determined by its function. When used with self-tapping screws,
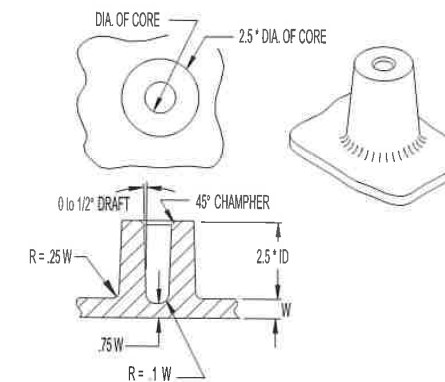

Figure 10.27 Standard recommended boss design. Though an inside draft of one half degree is suggested to assist with ejection, a zero degree draft is often preferred for functional purposes.

the inside diameter is commonly the pitch diameter of the screw. The outer diameter is approximately 2.5 times the inside diameter. This is a compromise between providing strength and minimizing the thickness of the boss. If a boss is too thick it could cause voids. As a self-tapping screw is inserted, it might break through to the void and thus reduce the pull-out strength. Outside diameters may vary slightly depending on screw design and the plastic material. Medium-to-low modulus materials can use a thread-forming screw. A variety of specially designed screws are available that will reduce the hoop stress on the boss. Very high modulus materials require a thread cutting screw because the strain from a thread forming screw would cause the boss to crack. There are a number of companies that provide specially designed thread forming and thread cutting screws for plastics. Each design has been developed for different material moduli and each company has its own design recommendations for inside and outside boss diameters. Some of these specialized screws are the Trilobe® from Continental Screw Company and the Hi-Lo from Elco® Industries.

The height of the boss should be no more than 2.5 times the inside diameter. This is to minimize the potential of deflection of the core forming the inside of the boss as well as filling, packing, and ejection problems. The intersection of the boss and the primary wall should have a radius that is 25% of the primary wall thickness, with a minimum radius of 0.25 mm. The intersection of the inner wall of the boss and the base should have a radius that is 10% of the primary wall thickness. The connecting wall thickness at the inside base of the boss should be 75% the primary wall thickness. A 45-degree chamfer should be placed on the inner top edge of the boss. A minimum of a 1/2-degree draft should be placed on the outside walls of the boss. Draft on the inside of the boss is also desirable, but it is often not used because it might compromise the assembly with a screw or a press fit. As a result, a stripper sleeve is generally required to assure ejection of the boss.

.It should be expected that the relatively large cross-section at the intersection of the boss and the primary wall will normally result in a sink, or gloss variation, on the primary wall surface opposite the boss. An alternate design for reducing the material volume at the intersection of the boss to the nominal wall may be used (see Fig. 10.28). Even though this design reduces potential for sinks and voids, it will
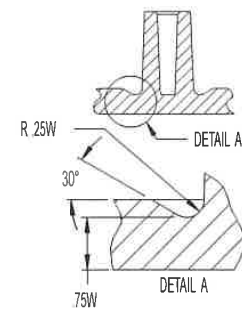

Figure 10.28 Alternate boss design for reducing the material volume at the intersection of the boss and the primary wall.

require an additional mold insert to form the depression that will increase the cost of the tooling.

10.3.4   Corners, Fillets, and Radii

Sharp corners should be avoided because most plastics are notch sensitive. This is particularly true with an inside corner that will act as a stress concentrator under load. Stress concentration will increase with the ratio of the corners fillet radius to the wall thickness. A preferred minimum radius-to-wall thickness (R/T) ratio would be 0.5. The sharper the radius, the more brittle the material will appear when loaded. In order to achieve this, the opposing wall surface must be radiused in order to avoid a thick region at the intersection.

Figure 10.29 illustrates the proper design of corners in a plastic part. This design provides for a constant wall thickness throughout the corner region. The inside radius should be a minimum of 0.5 (50%) the thickness of the primary wall. The outside radius should be the inside radius plus the wall thickness. This assures that the wall thickness is constant throughout the corner. (Note: The larger the inside radius the better.)

For example: Given a part having a corner with a primary wall thickness of 2.0 mm. The minimum inside radius of the corner should be 1.0 mm and the outside radius of the corner would be 3.0 mm.

This design maintains a uniform wall thickness in the corner. It is preferred that the inside radius be as large as possible. A larger radius can both improve the part structurally and reduce expected warpage developed from unbalanced cooling in corners. It may also improve filling in some circumstances, particularly with fiber-filled materials. In some cases, sharp corners are used for aesthetic purposes or to accommodate a mold machining issue. High loads and stresses should be avoided in these regions.
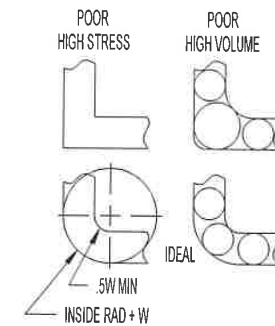

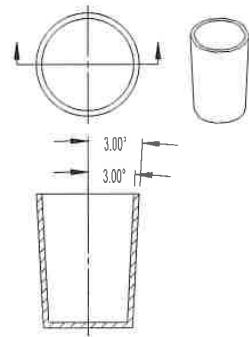Figure 10.29 Proper design of corners in a plastic part.

Figure 10.30 Draft on the inside and outside surfaces of a part should be equal and parallel.

10.3.5   Taper and Draft Angles

Drafts or tapers are angles put on vertical walls of an injection molded part to provide easier ejection from the mold. These angles generally range from 1/8 degree to several degrees, depending upon the material, anticipated ejection problems, and product design requirements. Draft on inside and outside surfaces of side walls should be equal and parallel (see Fig. 10.30). The greater the draft, the less potential for ejection problems.

10.3.6   Undercuts and Holes

Undercuts in a part can significantly increase the difficulty of ejecting molded parts from a mold. Examples of undercuts include a hole in the sidewall of a part, an internal snap ring which is often used in container caps, and internal and external threads. These require the use of special mold features such as side cores, split cavities, collapsible cores, unscrewing devices, and stripper plates. These features add to the initial cost of the mold, its maintenance, and molding cycle time. If possible, avoid holes or undercuts that would require these special mold requirements. If a hole is placed in a wall that is approximately perpendicular to the direction of the molding machine's platen movement, then a simple core may be used to form the hole and can easily be extracted during normal mold opening with the use of simple ejection; however, if the hole is created on the sidewall of a part, which is not perpendicular to the direction of the molding machine's platen movement, then the core forming the hole must generally be extracted by some special means. This side action would normally be performed by a side core, or split cavity, that would require some special means to move it perpendicular to the normal mold opening direction.

Some holes in side walls can be designed such that side action is not required. A stepped parting line can be used such that the hole is created while eliminating an undercut being created (see Fig. 10.31). A minimum shutoff angle of 5 degrees allows
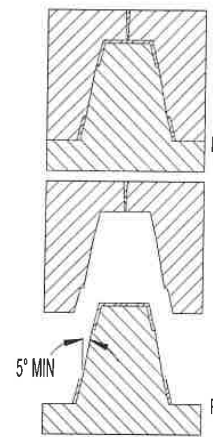
Figure 10.31 A stepped parting line can sometimes be used to create holes in the side walls of a part. This can eliminate the need for side action.

a clamp force to be developed between the mating mold halves to prevent flashing and helps to assure that the two halves will not scrape against each other during mold opening and closing. This method simplifies ejection by eliminating the need for side cores/slides.

Another example of the use of the stepped parting lines for creating undercuts and holes is the part shown in Fig. 10.32. The vents on the top of this part are designed to allow airflow and prevent objects from falling through the gaps. The intermeshing mold features that create these vents are illustrated in Fig. 10.33. A similar approach is used to create the vents on the back of the part. These intermeshing shutoffs create holes on both the top and side of the part without creating undercuts, thereby eliminating the need for special side action in the mold.

Holes should provide sufficient spacing to minimize ejection problems and weaknesses created by weld lines. If weld line strength is a particular concern, the holes may be spotted during molding and drilled as a postmolding operation. If the holes
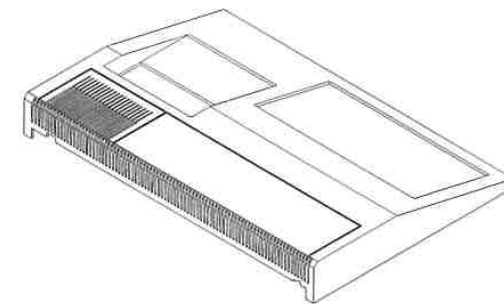

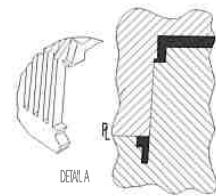Figure 10.32 Vents (undercuts and holes) created by use of a stepped parting line.

Figure 10.33 "Detail A" and the stepped parting line which creates the vents in Fig. 10.32.

are to be molded and will be structurally loaded, then the outer perimeter of the hole should be a minimum of 1 Diameter from the outside edge of a side wall (see Fig. 10.34A). For holes that serve a nonstructural need the spacing between them can be as little as twice the wall thickness of the part (Fig. 10.34B).

Some undercuts, like the snap ring on many container caps, may be stripped off the core forming it. When designing these snap rings, the designer must consider the design of the snap ring, the adjoining wall, and the material properties. The snap ring must be designed such that the part can be stripped, or slid off the core forming it without the snap ring, or the part, being damaged. This requires that the snap ring be designed with a reduced lead angle. A lead angle of 45 degrees is preferred (see Fig. 10.35). In addition, the sidewall must be able to be deflected during ejection. This requires consideration of the depth of the undercut, the diameter of the core, and the materials stress or strain limits. Because the properties of the material are not normally available at the relatively high temperatures when a part is ejected from the mold, designing the maximum undercut that can be stripped from the mold is often
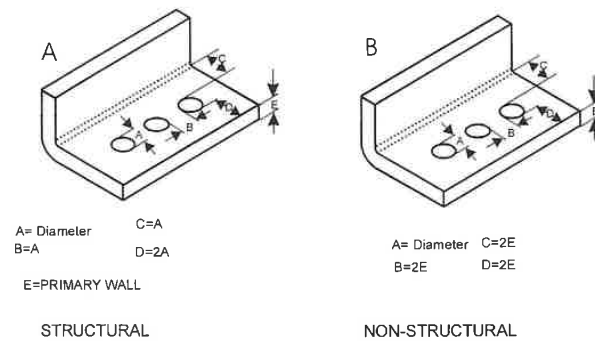

Figure 10.34 (A) Guideline for placement of holes in a part used for structural applications. (B) Guideline for placement of holes in a part used in nonstructural applications.
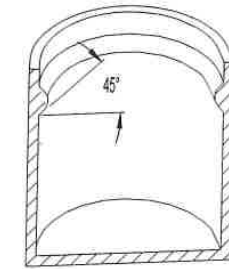
Figure 10.35 To facilitate stripping from a mold, a lead angle of 45 degrees is suggested.

based on the dynamic strain limits available for the room temperature material. This is generally a conservative design approach because the strain limits of most plastics are much higher at their ejection temperatures. The dynamic strain limit for a material is most representative of a materials characteristic under the rapid strain experienced by snap fits.

For example:
Determine the maximum height which can be used for a snap ring that can be used on a part which has an ID of 70 mm. The dynamic strain limit of the material is 3%.

$$\varepsilon = \frac{\Delta r}{r} \quad \Delta r = \varepsilon * r$$
$$\Delta r = 0.03 * 35 mm = 1.05 mm$$

The inside radius is shown as r, Δr is the maximum allowable change in radius; that would be the maximum allowable undercut. ε is the dynamic strain limit for the material. It should be expected that a stripper plate or stripper sleeve would be required to strip the part from the core.

The preceding is a simplified approach to designing an undercut. Prototyping is often required for verifying the design.

10.3.7 Gating and Process Considerations

One must always be cautious of the melt flow pattern during mold filling relative to a hole. There will be a weld line any time there is a hole in a part unless a disk-type gate is used inside the hole. This weld line will reduce strength and create potential negative cosmetic effects. Look to position the weld line strategically relative to these issues.

Whenever an undercut must be stripped from a core, be sure that there are no weld lines across the undercut. The high stress developed during ejection could cause the part to fail. The special core or cavity components, required to create these holes,

or undercuts, are generally separate mold inserts that require movement. This required movement, and the fact that these parts are not integral with the primary mold plates, result in these mold components being less rigidly anchored; therefore, when these mold cores become relatively long they are more susceptible to undesirable deflection created from the high melt pressures developed during mold filling and pack stages. This deflection can affect mold filling, part geometry, and development of flash problems. When selecting a gate position in a cavity that includes these components, one should consider the resulting applied forces. It may be desirable to position the gates away from these features to reduce the pressure acting on them.

10.3.8   Cores

Molded parts may include features created by tall cores, such as the inside of a boss, or may be formed entirely from a tall core, such as a test tube or pail. These tall cores can present a number of potential problems. High pressures can deflect the core causing variations in wall thickness and disruption of the expected fill pattern. For this purpose, the part should be designed such that the maximum height of the core forming it is 2.5 times the core outer diameter. Even a core that is center gated, where the melt is expected to flow symmetrically down its sides, can deflect. This can be caused by slight changes in melt conditions (shear and temperature effects) developed in branching runners or other factors, such as nonuniform cooling, that create an imbalance in the mold filling. This unbalanced filling will continue into the packing stages. The filling and packing imbalance causes a pressure variation from one side of the core to the other. Even 300-mm diameter cores used for molding a pail can deflect from this imbalance. Neither the melt variation created in a branching runner, nor its effect on a deflecting core, is realized with today's more common 2.5-D mold filling simulations.

Several designs have been used to help reduce core deflection. Figure 10.36A illustrates a design that incorporates a feature that has a self-centering effect. Here, if the melt leads over one side of the core and begins to deflect it, the flow path near the gate feeding this leading flow will be restricted while opening up the flow path on the opposing side. This local, bending-induced variation in wall thickness will potentially correct for the initial core deflection.

Figure 10.36B illustrates a second design where the base of a molded cup-shaped part is thickened relative to the side walls. If there is any imbalance developed out of the gate, the leading melt front will hesitate when it reaches the thinner side walls, allowing the slower flow fronts to catch up.

A part with a core gated on its side will be particularly sensitive to core deflection. In addition, this side gating scheme can result in severe weld lines or gas traps opposite the gate location. With edge gating, the creation of weld lines and gas traps can develop in even relatively short cores. A weld line might appear in a round core where the height is only 50% of the core's diameter. A gas trap during mold filling could appear in a core where the height is as little as 75% of the core diameter.
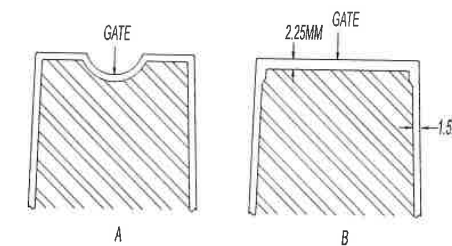
Figure 10.36 Two concepts of self-stabilizing cores.

10.3.9   Avoid Picture Frame Features

It is called a *picture frame* when the perimeter of the part is thicker than the interior (Fig. 10.37). This can create numerous problems:

- Gating in the thicker perimeter will result in a racetrack effect, where the melt quickly follows the path of the thicker perimeter and moves slowly through the thinner center region during mold filling. This can result in a gas trap or a severe weld line in the thinner region.
- Gating in the thinner center will result in voids and sinks in the thicker perimeter.
- In either gating location, the thicker perimeter will shrink more than the center region, which will cause the part to warp in a bowl-, or oil can–, like shape.

10.3.10 Integral Hinges

Integral hinges are very thin features molded into a part that provide local flexibility. These are commonly used in plastic containers and directly connect a box and its
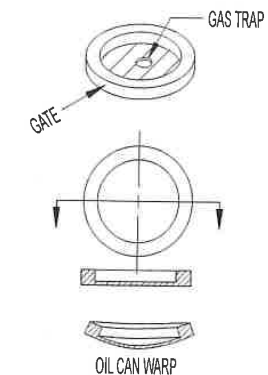

Figure 10.37 "Picture frame" part design. This type of design can result in numerous mold filling and warping problems.
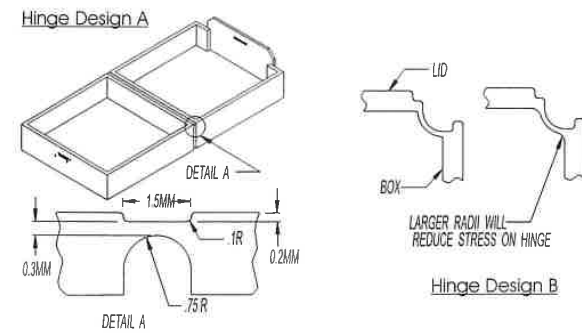
Figure 10.38 Common designs for integral hinges.

lid (Fig. 10.38). Although most commonly used with polypropylenes, integral hinges can be used with most of the more ductile plastic materials. Integral hinges made of polypropylene can last hundreds of thousands of cycles when correctly designed and molded. The strength and durability of the integral hinge comes from the high polymer orientation gained when material is forced to flow under high stresses across this thin feature. Poor flow or weld lines along the hinge will result in its premature failure.

There are many design variations for the integral hinge. They are generally very thin and short continuous strips joining the two parts sections to be hinged brackets (example: box and lid). Thickness is commonly only 0.3 mm with a width of less than 1.0 mm (see Fig. 10.38, Design A). These hinges can generally be bent in a 180+ degree arch and allow for a very small radiused closer at the hinge. A thicker noncontinuous hinge can also be used. These are more like straps and are longer to allow for deflection (see Fig. 10.38, Design B). These normally require a larger radius when closed.

One must be very careful when gating and molding the thinner integral hinge. With a container and lid combination, a gate is normally placed in the larger container. The gate should be placed away from the hinge such that it is the last place for the melt to reach. The melt should approach perpendicular to the hinge so that the flow front and the hinge are parallel. These strategies will help to avoid any hesitation of the melt front as it hits the restricted hinge. If the melt front reaches the hinge, while other regions are still being filled, it can hesitate and potentially freeze off.

An additional potential problem with integral hinges is packing of the cavity regions on the opposite side of the hinge from the gate. During packing, the material in the hinge will quickly freeze off while material in the thicker lid is still molten. Control of packing is lost and the lid can be expected to shrink differently than the container portion and may include undesirable sinks, voids, or loss of surface finish. If a gate is placed on both sides of the hinge to improve packing, extreme caution needs to be taken that the resulting weld line between the two gating locations does not occur at the hinge.

10.4   Sample Part Design

Given a simple manifold block to be produced from steel (see Fig. 10.39). The functional components are the central hole, which provides passage for low pressure air, and the four corner holes to position bolts for assembly. The only real structural requirement is that the part must withstand the compressive load of 70 pounds from each of the four assembly bolts. When produced from steel, the outer boundaries of the part would be machined from a solid block of steel (top figure) and then the five holes would be drilled (bottom figure). The process of machining steel is a subtractive process where it is desirable to minimize the amount of material to be removed in order to keep costs down. Coring out the unused material around the holes would simply add to the cost of machining the metal part. This additional step would only be done if there were some other requirements, including weight reduction.

When designing an alternate plastic part design, the designer should no longer be treating the part as something that is to be machined. The designer should focus on the functional component of the part, which is in this case the five holes, and build the part as though it is an additive process. During this building stage, the designer should be addressing the functional requirements while keeping in mind the objective to maintain a minimum wall thickness.

First, the five holes are created individually using thin features, which in this case are tubular structures (Fig. 10.40). The center hole has no real structural requirement and is kept relatively thin. The four mounting holes must withstand a compressive force (F) of 70 pounds each. Given the inside diameter (ID) of 0.25 in, the minimum outside diameter can be calculated. Applying a safety factor of 4:1, each boss must withstand a compressive force of 280 pounds. Given that the plastic material, which
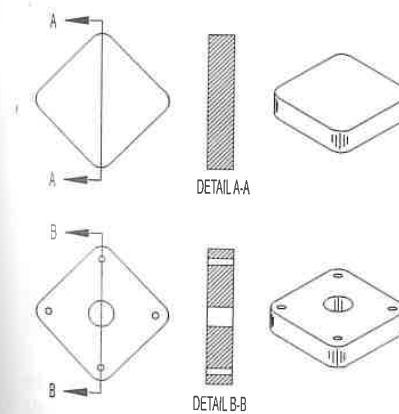

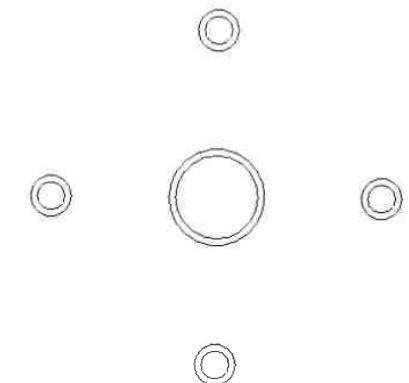Figure 10.39 Manifold block machined out of steel.

Figure 10.40 Functional components of manifold block in Fig. 10.39.

is to be used, has a compressive yield stress (σ) of 10,000 psi, the minimum outside diameter (OD) is found by:

$$\sigma = \frac{F}{A}$$

$$\sigma = \frac{F}{\left(\dfrac{\pi(OD)^2}{4} - \dfrac{\pi(ID)^2}{4} \right)}$$

$$OD = \sqrt{\frac{4\left(\dfrac{F}{\sigma} + \dfrac{\pi(ID)^2}{4} \right)}{\pi}}$$

$$OD = 0.313\,\text{in}$$

The minimum required wall thickness of boss is found to be 0.032 in.

As the four mounting holes, or bosses, act to locate the center air passage, they must be connected to it. Figure 10.42 illustrates one approach in providing this connection. Here a single thin wall connects each of the mounting bosses to the center air passage. At this stage all of the functional components have been included and combined into a single part. If more stiffness is required, ribs, or gussets might be added along the connecting walls. These are kept short and placed on either side of the wall to maintain symmetry to minimize potential for warpage (Fig. 10.43).
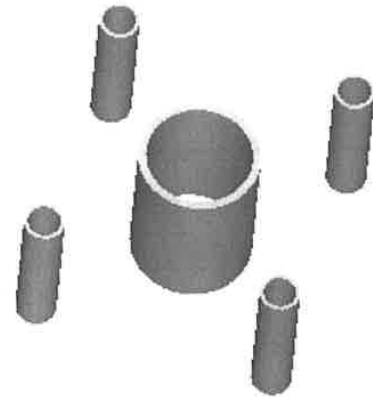

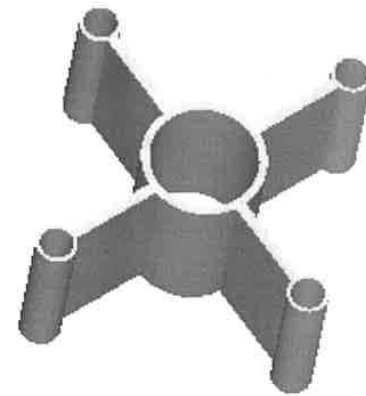Figure 10.41 Plastic design applied to functional components.

Figure 10.42 Tying the functional components of the manifold together to form a functional part.

Wall thicknesses must now be determined. As the only real structural requirement is satisfied with the 0.032-in thickness of the boss walls, all other walls could be the same or less. The 0.032-in thickness, however, is already quite thin. An injection mold filling simulation might be required if the designer wants to consider using a wall this thin. Once a wall thickness has been found for which the part can be safely filled (including consideration of the runner and gating system), all wall sections including the flow passage, the four connecting walls, and the ribs that support these walls could be designed to be the same thickness. The potential is that sinks or voids will appear at the intersection of any of the wall junctions. The only concern might be structural because this is not a cosmetic part. To minimize the risk, four gates could be positioned on the interior of the air channel, opposite the location of the four walls connecting to the boss. This would feed melt directly into the relatively thick intersecting wall locations that would act as flow leaders. These are all directly linked in this design. The result is that during the packing stage of the molding cycle, melt can be continually fed to these regions to compensate for the shrinking material. The gate would actually be slightly offset from the connecting wall so that it would not jet during initial injection.

Radiuses of 25% of the wall are added at the intersection of all walls to reduce stress at these intersections. The radiuses will increase the volume of material at the intersections, increasing the potential for voids, but the impact is again minimized due to the location of the gate and the connection of all of these regions to improve packing.

A variation of this design would be to thin the connecting four wall sections and their associated supporting ribs. The concern of varying wall thickness causing residual stress and warpage should not be a concern in this part. Unlike a thinned rib placed on the side of a thicker wall, a bending moment is not created. In this part, the wall thickness variation might cause the heights of the center air passage and the four bosses to shrink in height more than the thinner connecting walls. This might create a residual stress, but due to their relative relationship and structure, would be unlikely to cause the part to warp. If the boss height shrinks by more than the connecting walls, then the stress during assembly may be diverted to the taller walls, which might engage connecting components. This could be addressed by reducing the height of the walls in the design stage. This would assure that the bosses take up the stress, and a close interface of the center air passage is maintained.

Further concerns with this design variation might include feeding material from the gate, through a thin wall to the thicker walls of the boss. This is somewhat alleviated by the fact that the melt flow channel along the connecting rib is created by the intersection of actually four walls (the top and bottom of the connecting wall and the two sides of the supporting ribs). This increases the channel size and should improve the ability to feed material to the boss during packing stage.

Throughout the design, ejection from the mold must be considered. With either of the preceding designs, the cavity could be positioned such that the parting line of the mold is along the center plane of the part, with equal portions of all components formed in both the stationary (cavity) half and the movable (core) half of the mold. The inside diameters could similarly be formed from cores projecting from either
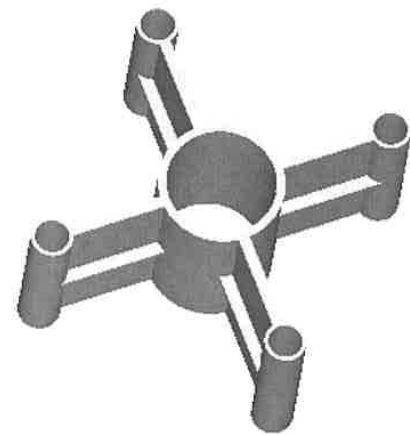
Figure 10.43 Addition of ribs and draft to the molded plastic manifold for increased stiffness and ease of ejection.
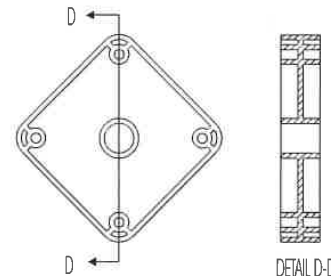

Figure 10.44 Variation of the manifold design that can be used for injection molding.

mold halves. Drafts are added to facilitate ejection and are located on the inside and outside walls of the center flow channel, the four bosses and the four connecting walls. To assure that the part sticks to the ejection half of the mold during mold opening, the draft angle applied to this half of the mold should be less than the stationary half. Stripper sleeves should be used to eject the part from each of the five cores creating the inside diameter of the holes.

Give the development of this part to five different designers, and five different designs will develop. Another variation of this design is shown in Fig. 10.44. The functional components are again created with thin-walled features and then connected. In this design a frame is developed around the perimeter of the part. Supporting gussets, or ribs, connect the four bosses to the outside wall.

10.5   Estimating Part Costs

Due to the high cost of an injection mold, the cost of injection molded parts are significantly effected by production volume. It is rare that injection molding can be justified if production is expected to be less than 10,000 parts per year. Alternate processes such as vacuum forming, pressure forming or machining need to be evaluated for smaller production levels.

Part cost is a function of direct, packaging, shipping and indirect costs. Direct costs include materials, direct labor plus any dedicated mold or other tooling that is re-

quired to produce the parts. Indirect costs include the elements of the business that are required for its operation, but don't directly produce a product. These include the building expenses, equipment, and support staff. Support staff can include engineering, sales force, company management, maintenance personnel, etc.

Direct costs can be found as follows:

1. Determining the volume of the molded part and the material cost per volume.
2. Direct labor is based on knowing how many parts are produced per hour, the hourly rate associated with any direct labor (including overhead), and the number of laborers required for a particular job. If a job is to run fully automatic, there is still a fraction of a labor cost assigned as someone must be responsible for handling the molded parts, loading material, etc.
3. Dedicated tooling/equipment costs. This cost is very sensitive to production volume. In addition, this cost can be significantly influenced by the risk and business strategy of the company as effected by the duration over which investments are to be amortized.

Given a case where a part requires a mold that costs $200,000. Of these parts 100,000 are to be produced per year over 5 years (total of 500,000 parts). The cost of the mold is commonly amortized over some period of time that is dependent on the company's business strategy.

The company could choose to spread the cost of the mold over the 5 years and 500,000 parts. This would result in a cost of approximately $0.40/part ($200,000/500,000 parts). If they take 5 years to pay for the mold, however, the $200,000 used to purchase the mold is a long-term debt that must reflect an interest cost to them. Even if the company had the cash to pay for the mold, this is money lost to them which could have been providing capital gains through investments. Either way a cost for use of this money must be factored. The cost therefore will be something more than $0.40/part and will be presented later.

As with any purchase that is spread over time, the advantage is that the short-term costs are minimized. By this means the cost of the product is also minimized. The further advantage of this is that the cost to the customer is lowered. The problem with spreading the cost over an extended period of time is that there is an increased risk to the manufacturer. The risk is that the product does not sell and production is terminated after only 1 year. As a result the manufacturer has only recovered $40,000 from the $200,000 cost of the mold. He will therefore lose $160,000.

The alternative is to try to amortize the mold cost over one year. In this case the cost of each part during the first year would have to be $2.00 versus only $0.40. The disadvantage here is that your part will cost more than someone who elected to amortize their parts through 5 years; however there are a number of benefits to this method:

1. If you succeed in selling your part for the $2.00 during the first year, then your mold has been paid for. During the following 4 years of production you can either maintain your prices and make a hefty profit or drop your price to combat any competition.

2. If the product does not sell after the first year, you will have recovered your cost and will not take a loss.

3. Since you only took one year to pay for the mold, the cost of money will be significantly less.

In determining the cost of a product, indirect costs are commonly factored into an hourly rate that is developed around the primary piece of manufacturing equipment to be used to produce the parts. In the case of an injection molded article, an hourly rate is assigned to each of the injection molding machines according to its cost. This is typically reflected in the size (tonnage rating) of the machine. These costs will vary significantly from company to company and even plant to plant. Example of costs for various injection molding machines are as follows.

100 Ton = $30/hr
200 Ton = $34/hr
300 Ton = $38/hr

An example of calculating part cost follows. This example does not include the cost of packaging or shipping.

Number of cavities	= 16
Production cost	= 50 $/hour
Direct labor cost	= 8 $/hour
Annual production	= 5,000,000 parts/year
Number of year for amortizing tooling	= 3
Molding cycle	= 16 seconds
Total number of payments during the time that the mold will be amortized	= 36
Mold cost	= $100,000
Material cost	= 2.0 $/lb
Number of annual payments on the mold	= 12
Part weight	= 10 grams
Production Yield	= 95%

Material Cost per Part Determine material cost per part. Given that the material cost is given in dollars per pound ($/lb), and part weight is given in grams(g), we must convert part weight from grams to pounds when determining material cost per part:

$$\text{Material cost per part} = \left(\frac{\text{Part Weight}}{454}\right) \cdot \text{Material Cost}$$

Material cost per part = $0.0441

Production Cost First, determine the number of parts produced per hour.

$$\text{Parts per hour} = \frac{3600}{\text{Molding cycle}} \cdot \text{Number of cavities}$$

Parts per hour = 3600

Second, determine production cost for each part:

$$\text{Production cost per part} = \frac{\text{Production Cost} + \text{Direct labor}}{\frac{3600}{\text{Molding cycle}} \cdot \text{Number of cavities}}$$

Production cost per part = 0.0161

Mold Cost per Part First, determine the total cost of the mold (mold cost + interest)

$$\text{Total cost} = \text{Mold cost} \cdot \text{Total number of payments} \cdot$$

$$\left[\frac{\left(\frac{\text{Interest rate}}{\text{Num annual pay}}\right)\left(1 + \frac{\text{Interest rate}}{\text{Num annual pay}}\right)^{\text{Total number of payments}}}{\left(1 + \frac{\text{Interest rate}}{\text{Num Annual pay}}\right)^{\text{Total number of payments}} - 1}\right]$$

Total cost = $116,162

Second, determine the mold cost per part: Total cost of the mold divided by the total production over the years for which the total mold cost will be amortized.

$$\text{Tooling Cost per Part} = \frac{\text{Total Cost}}{\text{Annual Production} \cdot \text{Number of Years Amoritized}}$$

Tooling Cost per Part = 0.0077

Total Cost = (Material Cost + Production Cost + Mold Cost) × Yield

$$\text{Total part cost} = (\text{Material cost per part} + \text{Production cost per part} + \text{Tooling cost per part}) \cdot \text{Yield}$$

Total Part Cost = $0.0645

The preceding example applies the 5% spoilage to the total cost. In some cases a different spoilage might be applied to material cost, production cost, and materials.

References

1. Wagner, A. H. Yu, J. S., and Kalyon, D. M., Orientation and Residual Stress Distribution in Injection Molded Engineering Plastics, Proceedings of the Society of Plastics Engineering 47th Annual Technical Conference, SPE, Brookfield Center, CT, 1988, 303–307.
2. Jaros, B. G., Shrinkage of Glass-Filled Parts as Developed by Radial Flow, Proceedings of the Society of Plastics Engineering 5th Annual Technical Conference, SPE, Brookfield Center, CT, 1991, 2646–2648.