CHAPTER 6.2

INJECTION-MOLDED THERMOPLASTIC PARTS

Thermoplastic materials are synthetic organic chemical compounds of high molecular weight that soften or liquefy when they are heated and solidify when they are cooled. When cooled, they are relatively tough and durable and suitable for a wide variety of product applications.

THE PROCESS

These materials are formed to specific shapes by injecting them when heated into a mold from which they take their final shape as they cool and solidify. The plastic material normally is received by the molder in granular form. It is placed in the hopper of an injection-molding machine, from which it is fed to a heated cylinder. As the granules heat in the cylinder, they melt, or plasticize. A typical melting temperature is about 180°C (350°F), although this varies with different materials and molding conditions. The mold, usually of steel, is clamped in the machine and is water-cooled. A plunger forces plasticized material from the cylinder into the mold. There it cools and solidifies. The mold is opened, and the molded part with its attached runners is removed. The process, with the occasional exception of part removal, is automatic. It requires about 45 s/cycle, more or less, with most of that time being devoted to cooling of the material in the mold. Very high pressures, on the order of 70,000 kPa (10,000 lbf/in^2) or more, are required during injection. Figure 6.2.1 illustrates the injection-molding process.

TYPICAL CHARACTERISTICS OF INJECTION-MOLDED PARTS

Injection molding is particularly advantageous when intricate parts must be produced in large quantities. Although there are limitations, as discussed below, generally the more irregular and intricate the part, the more likely it is that injection molding will be economical. In fact, one major advantage of the injection-molding method is that one molded part can replace what would otherwise be an assembly of components. (See Fig. 6.2.2.) In addition, color and surface finish often can be molded directly onto the part, so that secondary finishing operations are not necessary.

Injection-molded parts are generally thin-walled. Heavy sections and variable wall thicknesses are possible, though they are normally not recommended.

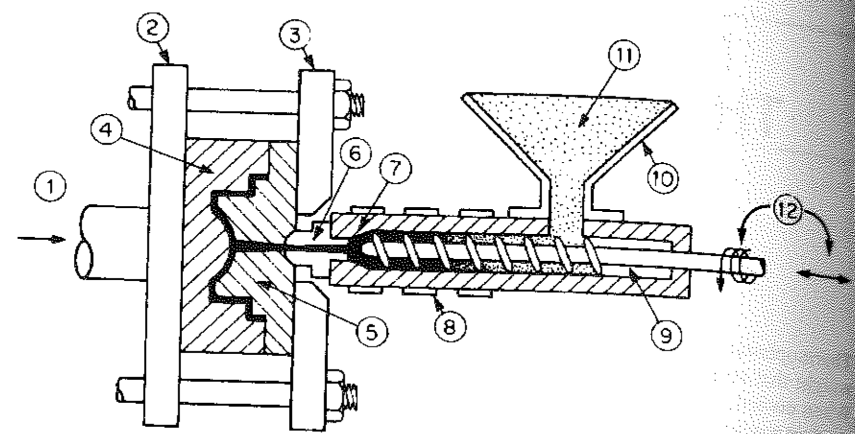

FIGURE 6.2.1 Injection molding. (1) Mold-clamping force. (2) Movable mold platen. (3) Fixed platen. (4) Cavity half of mold. (5) Force half of mold. (6) Nozzle. (7) Cylinder. (8) Electric band heaters. (9) Reciprocating screw. (10) Hopper. (11) Granulated-plastic material. (12) Rotary and reciprocating motion of screw.

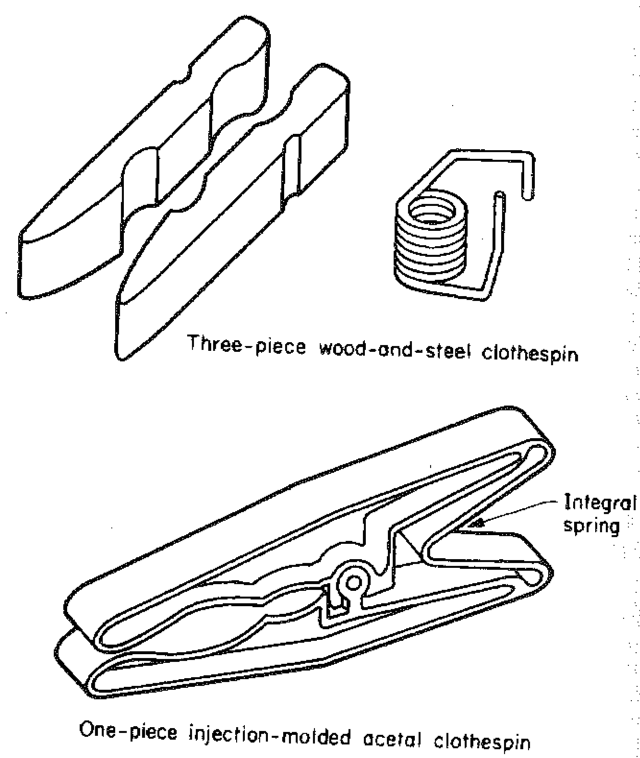

FIGURE 6.2.2 One-piece injection-molded clothespin. Two sections, connected by an integral spring, are assembled together at the pivot point after molding. (*Courtesy Celanese Plastics & Specialties Co.*)

Because thermoplastics are generally less strong than metals, they are more apt to be found in less highly stressed applications. Housings and covers are common uses rather than, for example, frames and connecting rods. However, thermoplastic materials are gradually being developed with better and better strength characteristics and are increasingly finding themselves used for moving parts and in more structural applications. The "engineering plastics," nylon, polycarbonate, acetal, phenylene oxide, polysulfone, thermoplastic polyesters, and others, particularly when reinforced with glass or other fibers, are functionally competitive with zinc, aluminum, and even steel.

Effects of Shrinkage

All thermoplastics exhibit shrinkage on cooling and solidification. Table 6.2.1 summarizes the extent of this shrinkage for common materials. In addition to effecting a reduction in most dimensions, shrinkage of plastic material causes various irregularities and warpage in the molded part. The most common such defect is the sink mark, or surface depression, opposite heavy sections. (See Fig. 6.2.3.)

TABLE 6.2.1 Shrinkage Rates of Common Thermoplastics upon Solidification in Mold

Thermoplastic	Percent
Acetal	2.0–2.5
Acrylic	0.3–0.8
Acrylonitrile butadiene styrene	0.3–0.8
Nylon	0.3–1.5
Polycarbonate	0.5–0.7
Polyethylene	1.5–5.0
Polypropylene	1.0–2.5
Polystyrene	0.2–0.6
Polyvinyl chloride, rigid	0.1–0.5
Polyvinyl chloride, flexible	1.0–5.0

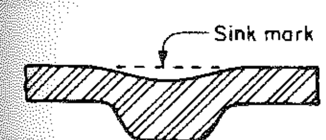

FIGURE 6.2.3 Typical sink mark opposite a heavy section.

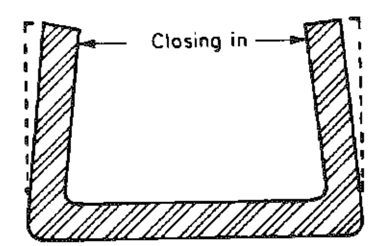

FIGURE 6.2.4 Shrinkage of plastic material on cooling causes the closing in of U-shaped sections.

Another common effect is the closing in of a U-shaped cross section, particularly if there are reinforcing ribs. (See Fig. 6.2.4.) A third common effect of shrinkage is the

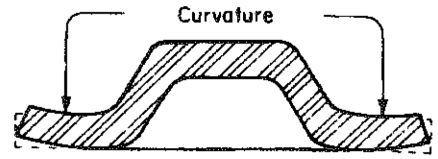

FIGURE 6.2.5 Curving of flat surfaces caused by shrinkage of material.

occurrence of curvature on a flat surface in the direction of a boss, protuberance, or added material. Figure 6.2.5 illustrates this problem.

ECONOMIC PRODUCTION QUANTITIES

Injection molding is a mass-production process. It is generally not applicable unless 10,000 or more identical parts are to be produced. The reason for this limitation is the necessity for constructing a unique mold for each part. Production must be large enough so that the mold cost can be amortized over the quantity manufactured. Even for smaller parts, molds can be costly, on the order of several thousands of dollars. For larger, intricate parts, they can cost tens or hundreds of thousands of dollars.

SUITABLE MATERIALS

A large number of suitable thermoplastics are available to the injection molder. Some of the more commonly used are polyethylene, polypropylene, polystyrene, polyvinyl chloride (vinyl or PVC), nylon, acrylonitrile butadiene styrene (ABS), and acrylic.

Because of the importance of injection molding to the commercial sale of thermoplastic materials, producers of these materials engineer them to be processible by injection molding. Physical properties and cost rather than processibility are normally the determining factors in the selection of materials for injection-molded parts.

Generally, the high-property engineering plastics are not as easy to mold as the commodity plastics such as polyethylene, polypropylene, and polystyrene. In addition, polyvinyl chloride, though low in cost and having very good physical properties, is more difficult to injection-mold than many other materials. Its prime drawback is a narrow temperature range between its melting and degradation points.

Table 6.2.2 lists common thermoplastics used for injection molding and indicates some of their properties, their cost, and their typical applications.

DESIGN RECOMMENDATIONS

Gate and Ejector-Pin Locations

The designer should consider the location of these elements because they can impair surface finish. Ejector pins usually can be located on the underside of a part if it has an outside and an underside. Gates can be located in a number of locations, as illus-

TABLE 6.2.2 Common Thermoplastics Materials Used for Injection Molding

Material	Tensile strength, MPa (lbf/in²)	Maximum service temperature, °C (°F)	Specific gravity	Approximate cost index*	Applications and remarks
Polystyrene	48 (7000)	82 (180)	1.04	1.0	Toys, containers
Polypropylene	34 (5000)	110 (230)	0.91	1.1	Housewares, appliances
High-density polyethylene	28 (4000)	125 (260)	0.96	1.1	Refrigerator parts, housewares
Low-density polyethylene	14 (2000)	88 (190)	0.92	1.3	Kitchen utility wear, toys
Acrylonitrile butadiene styrene (ABS)	48 (7000)	93 (200)	1.06	2.1	Housings, housewares; can be chrome-plated; handle grips; electrical plugs
Polyvinyl chloride, flexible	21 (3000)	82 (180)	1.40	1.0	Seals, electrical plugs, footwear components
Nylon	76 (11,000)	120 (250)	1.14	3.9	Bearings, gears, rollers
Styrene acrylonitrile	69 (10,000)	82 (180)	1.05	2.7	Kitchenware
Cellulosics	41 (6000)	70 (160)	1.22	4.0	Optical parts, tool handles
Polyacetal	69 (10,000)	90 (195)	1.43	3.3	Appliance parts, gears, etc.
Polycarbonate	62 (9000)	110 (230)	1.22	4.1	Portable-tool housings, automotive parts
Acrylic	69 (10,000)	78 (170)	1.18	2.0	Lenses, automotive trim

*1997 prices for large-quantity purchases; polystyrene = 1.0.

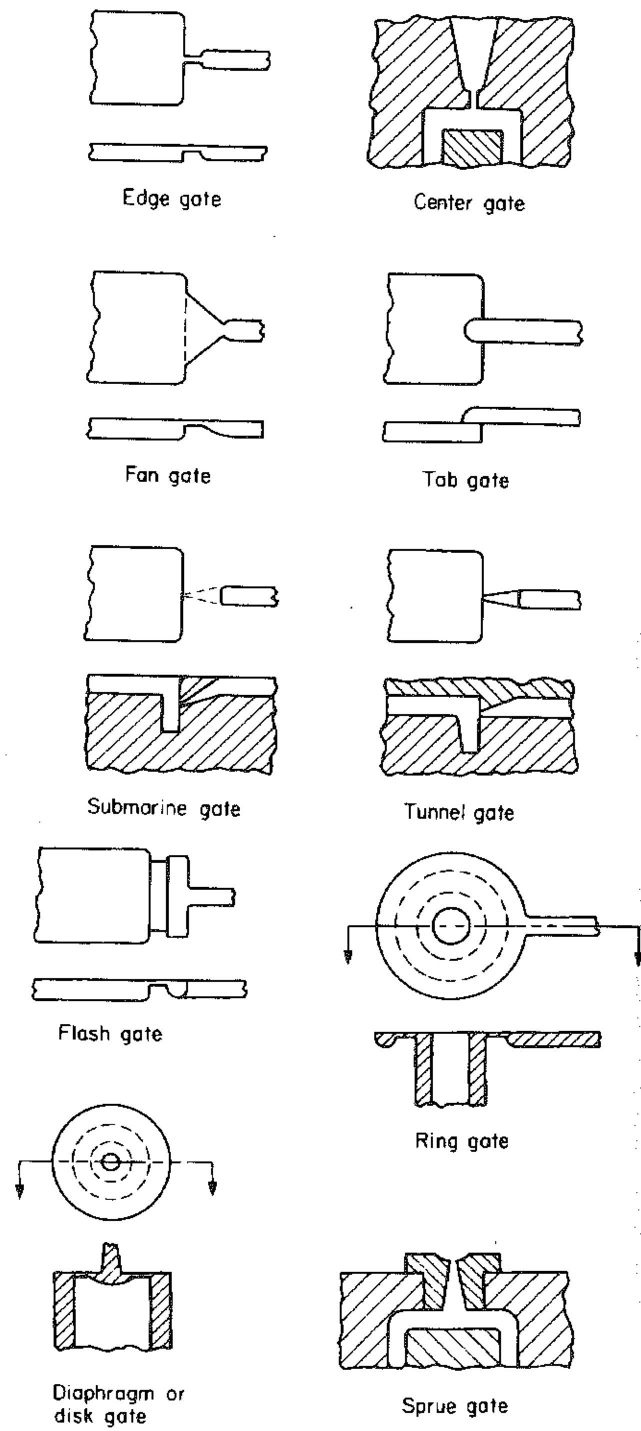

FIGURE 6.2.6 Various gating systems.

trated in Fig. 6.2.6. Center gating of round and cylindrical parts and near-center gating of other large-area parts are desirable for trouble-free mold filling.

Suggested Wall Thickness

Table 6.2.3 provides recommended normal and minimum wall thicknesses for common thermoplastics when injection-molded. Generally, thinner walls are more feasible with small parts rather than with large ones. The limiting factor in wall thinness is the tendency for the plastic material in thin walls to cool and solidify before the mold is filled. The shorter the material flow, the thinner the wall can be. Walls also should be as uniform in thickness as possible to avoid warpage from uneven shrinkage. When changes in wall thickness are unavoidable, the transition should be gradual, not abrupt. (See Fig. 6.2.7.)

TABLE 6.2.3 Suggested Wall Thicknesses for Various Thermoplastic Materials

Material	Short sections	Small sections	Average sections	Large sections
Acetal	0.6 (0.025)	0.9 (0.035)	1.9 (0.075)	3.2–4.7 (0.125–0.185)
Acrylic	0.6 (0.025)	0.9 (0.035)	2.3 (0.090)	3.2–6.3 (0.125–0.250)
Acrylonitrile butadiene styrene	0.9 (0.035)	1.3 (0.050)	1.9 (0.075)	3.2–4.7 (0.125–0.185)
Cellulose acetate butyrate	0.6 (0.025)	1.3 (0.050)	1.9 (0.075)	3.2–4.7 (0.125–0.185)
Nylon	0.3 (0.012)	0.6 (0.025)	1.5 (0.060)	2.4–3.2 (0.093–0.125)
Polycarbonate	0.4 (0.015)	0.8 (0.030)	1.8 (0.070)	2.4–3.2 (0.093–0.125)
Polyethylene				
Low-density	0.9 (0.035)	1.3 (0.050)	1.6 (0.062)	2.4–3.2 (0.093–0.125)
High-density	0.9 (0.035)	1.3 (0.050)	1.9 (0.075)	3.2–4.7 (0.125–0.185)
Polypropylene	0.6 (0.025)	0.9 (0.035)	1.9 (0.075)	3.2–4.7 (0.125–0.185)
Polystyrene	0.8 (0.030)	1.3 (0.050)	1.6 (0.062)	3.2–6.3 (0.125–0.250)
Polyvinyl chloride				
Flexible	0.6 (0.025)	1.3 (0.050)	1.9 (0.075)	3.2–4.7 (0.125–0.185)
Rigid	0.9 (0.035)	1.6 (0.062)	2.4 (0.093)	3.2–4.7 (0.125–0.185)

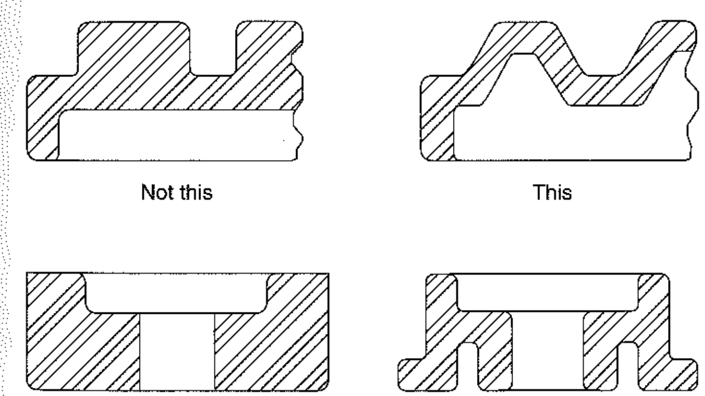

FIGURE 6.2.7 Maintain uniform wall thickness insofar as possible, and if changes in wall thickness are unavoidable, make them gradual rather than abrupt.

Holes

1. Holes are feasible in injection-molded parts but are a complicating factor in mold construction and part quality. "Knit" or "weld" lines adjacent to the hole often develop, and flashing also may occur at the edge of the hole. Figure 6.2.8 illustrates how weld lines are formed.
2. The minimum spacing between two holes or between a hole and sidewall should be one diameter. (See Fig. 6.2.9.)
3. Holes should be located three diameters or more from the edge of the part to avoid excessive stresses. (See Fig. 6.2.10.)
4. A through hole is preferred to a blind hole because the core pin that produces the hole can then be supported at both ends, resulting in better dimensional location of the hole and avoiding a bent or broken pin.
5. Holes in the bottom of the part are preferable to those in the side because the latter require retractable core pins.
6. Blind holes should not be more than two diameters deep. If the diameter is 1.5 mm ($^1/_{16}$ in) or less, one diameter is the maximum practical depth. (See Fig. 6.2.11.)
7. To increase the depth of a deep blind hole, use steps. This enables a stronger core pin to be employed. (See Fig. 6.2.12.)
8. Similarly, for through holes, cutout sections in the part can shorten the length of a small-diameter pin. (See Fig. 6.2.13.)
9. Use overlapping and offset mold-cavity projections instead of core pins to produce holes parallel to the die-parting line (perpendicular to the mold-movement direction). Figures 5.4.9 and 5.4.14 illustrate this approach, which is applicable to injection moldings as well as to die castings.

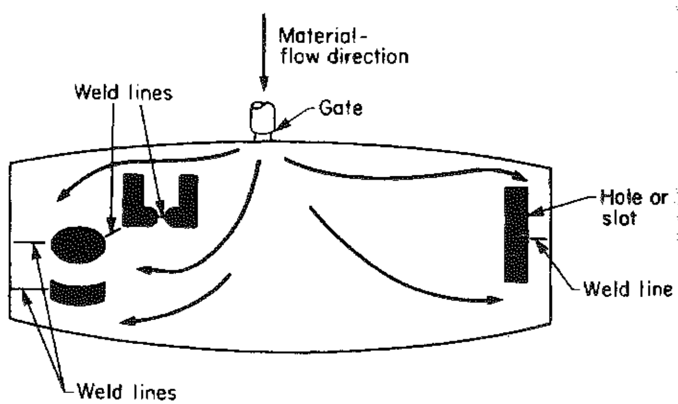

FIGURE 6.2.8 Weld or knit lines are caused when material flowing around hole core pins does not fuse together.

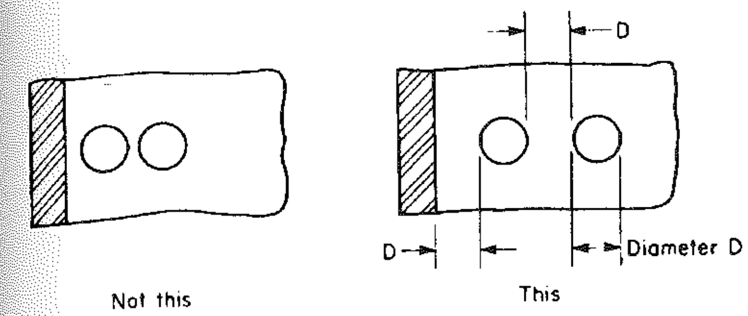

FIGURE 6.2.9 Minimum spacing for holes and sidewalls.

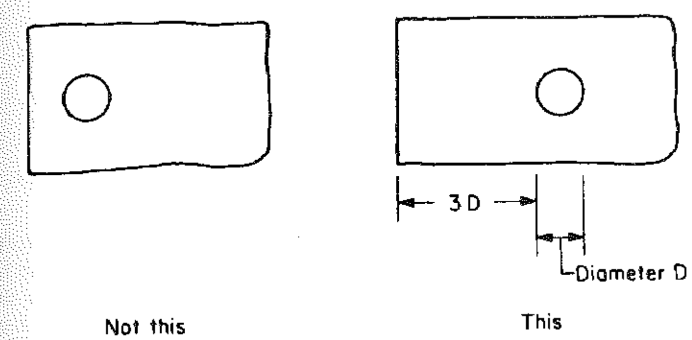

FIGURE 6.2.10 Minimum distance between a hole and the edge of the part.

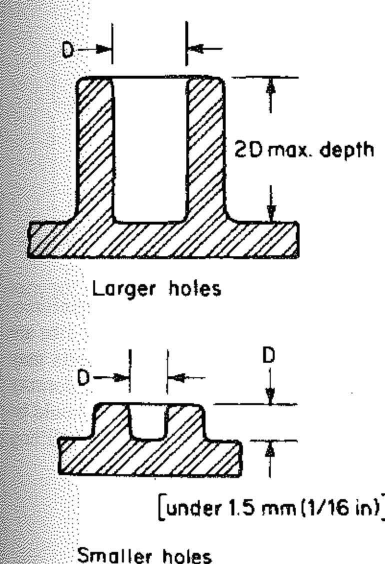

FIGURE 6.2.11 Recommended depth limits for blind holes.

Ribs

1. Reinforcing ribs should be thinner than the wall they are reinforcing to prevent sink marks in the opposite side of the wall. A good rule of thumb is to keep rib thickness to between 40% and 60% of the wall thickness.
2. Ribs should not be higher than 2.5 to 3.0 times the wall thickness.
3. Two ribs may be used, if necessary, to provide the extra reinforcement that would otherwise be provided by a high rib. The ribs should be two or more wall thicknesses apart.
4. Ribs should be perpendicular to the parting line to permit removal of the part from the mold.
5. Sink marks caused by ribs can be disguised or hidden by grooves or surface texture opposite the rib. (See Fig. 6.2.14.)

6. Ribs should have a generous draft of 0.5° to 1.5° per side.
7. There should be a radius at the base of 25% to 40% of the wall thickness.

See Fig. 6.2.15 for an illustration of most of these rules.

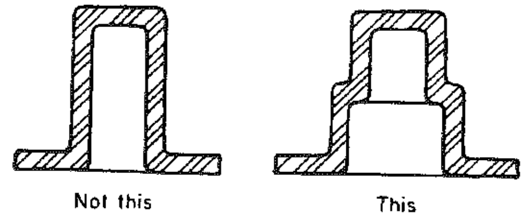

FIGURE 6.2.12 If a blind hole must be deep, use a stepped diameter.

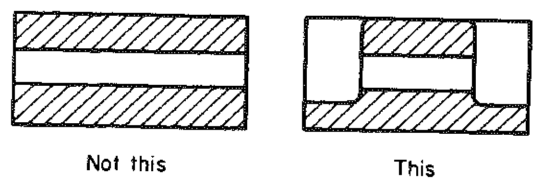

FIGURE 6.2.13 The improved design on the right provides better rigidity of the mold core pin.

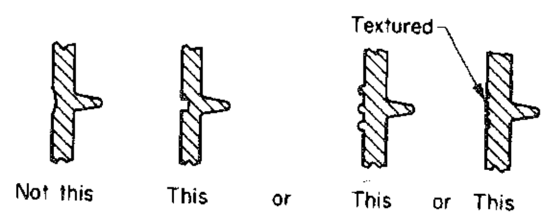

FIGURE 6.2.14 Methods for disguising sink marks.

Bosses

Bosses are protruding pads that are used to provide mounting surfaces or reinforcements around holes. The rules above for maximum height, draft, and radius of ribs are generally applicable to solid or hollow bosses. However, those holding inserts may require heavier walls. If large bosses are needed, they should be hollow for uniformity of wall thickness. Bosses in the upper portion of the mold can trap gasses and should be avoided, if possible. Locate bosses in corners, if possible, to aid material flow in filling the mold. If a detached boss is necessary, a connecting rib will aid material flow.

Undercuts

Undercuts are possible with injection-molded thermoplastic parts, but they may require sliding cores or split molds. (See Fig. 6.1.6.) External undercuts can be placed at the parting line or extended to the line to obviate the need for core pulls.

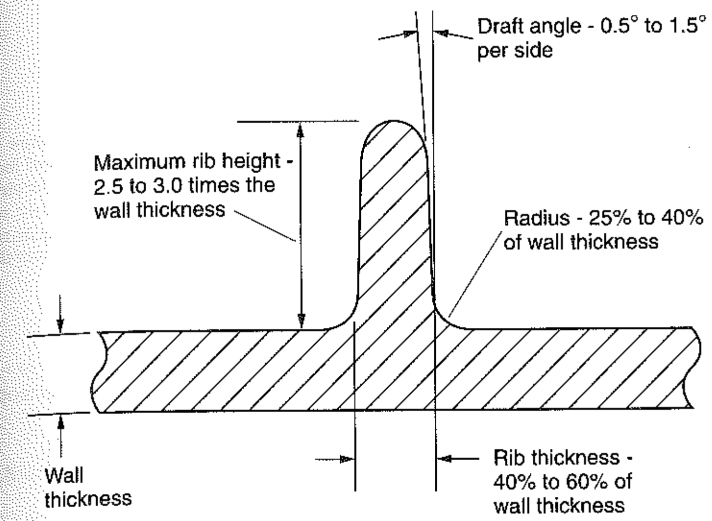

FIGURE 6.2.15 Design rules for reinforcing ribs and solid bosses.

Shallow undercuts often may be strippable from the mold without the need for core pulls. If the undercut is strippable, the other half of the mold must be removed first. Then the mold ejector pins can act to strip the part. Figure 6.2.16 shows the average maximum strippable undercut for common thermoplastics.

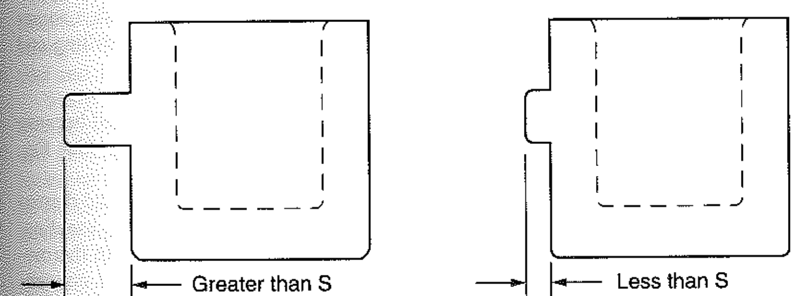

FIGURE 6.2.16 Allowable undercut for common materials.

Material	Average maximum strippable undercut S, mm (in)
Acrylic	1.5 (0.060)
Acrylonitrile butadiene styrene	1.8 (0.070)
Nylon	1.5 (0.060)
Polycarbonate	1.0 (0.040)
Polyethylene	2.0 (0.080)
Polypropylene	1.5 (0.060)
Polystyrene	1.0 (0.040)
Polysulfone	1.0 (0.040)
Vinyl, flexible	2.5 (0.100)

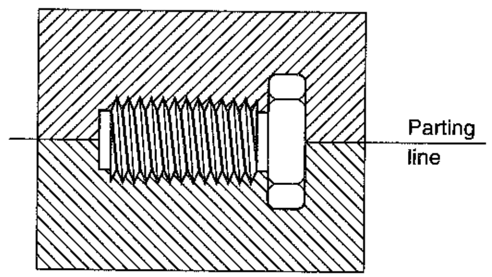

FIGURE 6.2.17 External screw threads can be molded without the need for a core pull if the threaded element is placed on the mold-parting plane. However, removal of flash from the threads may be required unless threads are excluded from the parting-line area.

Screw Threads

It is feasible, though a complicating factor, to mold screw threads in thermoplastic parts. Three basic methods can be used:

1. Use a core that is rotated after the molding cycle has been completed. This unscrews the part and enables it to be removed from the mold.
2. Put the axis of the screw at the parting line of the mold. This avoids the need for a rotating core but necessitates a very good fit between mold halves to avoid flash across the threads. This method is applicable only to external threads and generally leads to higher-cost parts unless the threads can be omitted in the area of the parting line. Often this is feasible. If this approach is not used, an extra operation probably will be required to remove parting-line flash from the threads. (See Fig. 6.2.17.)
3. Make the threads few, shallow, and of rounded form so that the part can be stripped from the mold without unscrewing. (See Fig. 6.2.18.) A coarse thread with a somewhat rounded form is preferred for all screw threads because of ease of filling and avoidance of featheredges even if it is removed by unscrewing.

Continuation of threads to the ends of threaded sections should be avoided because they create featheredges, make mold fit more critical, and promote flashing. (See Fig. 6.2.19.) If threads with strong holding power are needed, use metal inserts.

Internal threads can be tapped in almost all thermoplastics, and if the thread diameter is small [5 mm ($^3/_{16}$ in) or less], tapping is usually more economical than molding. Tapped or molded threads finer than 32 threads per inch are not practical with thermoplastics. Self-tapping screws are preferable to tapped or molded threads and a conventional screw.

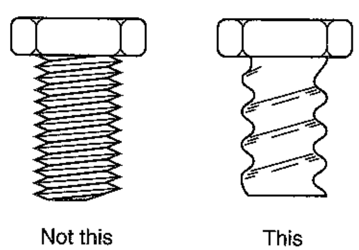

FIGURE 6.2.18 Shallow screw threads can sometimes be stripped directly from the mold.

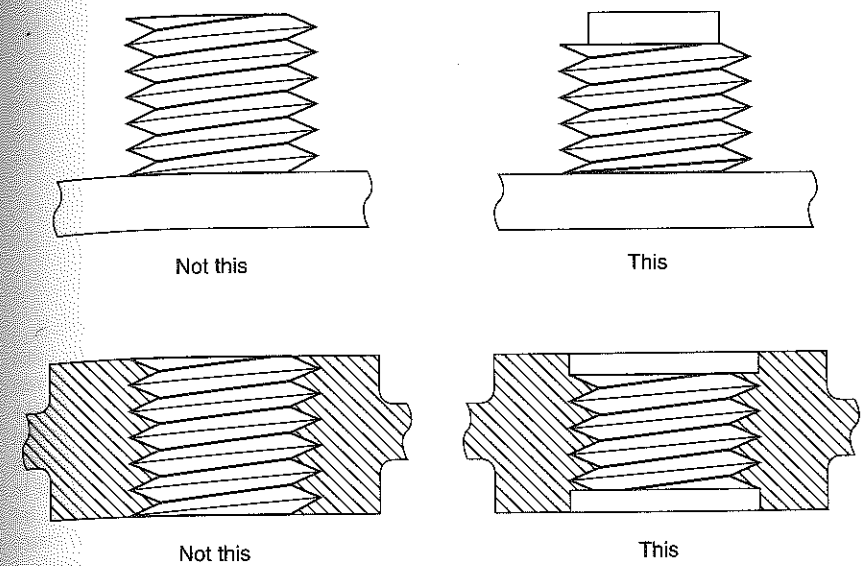

FIGURE 6.2.19 Screw threads should not extend to the ends of the threaded element.

Inserts

Inserts are useful and practical to provide reinforcement where stresses exceed the strength of the plastic material. Although they are economical, they are not without cost and should be used only when necessary for reinforcement, anchoring, or support.

Sharp corners should be avoided on the portion of the insert that is immersed in the thermoplastic. Figure 6.2.20 shows acceptable designs for the inserted ends of hooks and anchoring rods. Figure 6.2.21 shows unacceptable and acceptable designs and locations for inserted ends of screw-machine parts. Knurls on machined inserts should be relatively coarse to permit the material to flow into the recesses. There should be a smooth surface where the insert exits from the plastic.

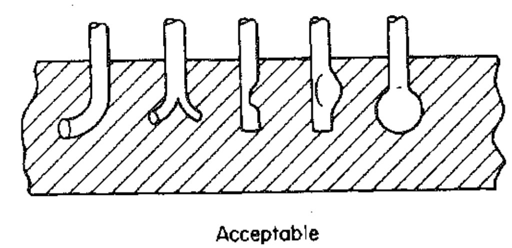

FIGURE 6.2.20 Recommended designs for the ends of hooks or rods to be inserted in plastic parts.

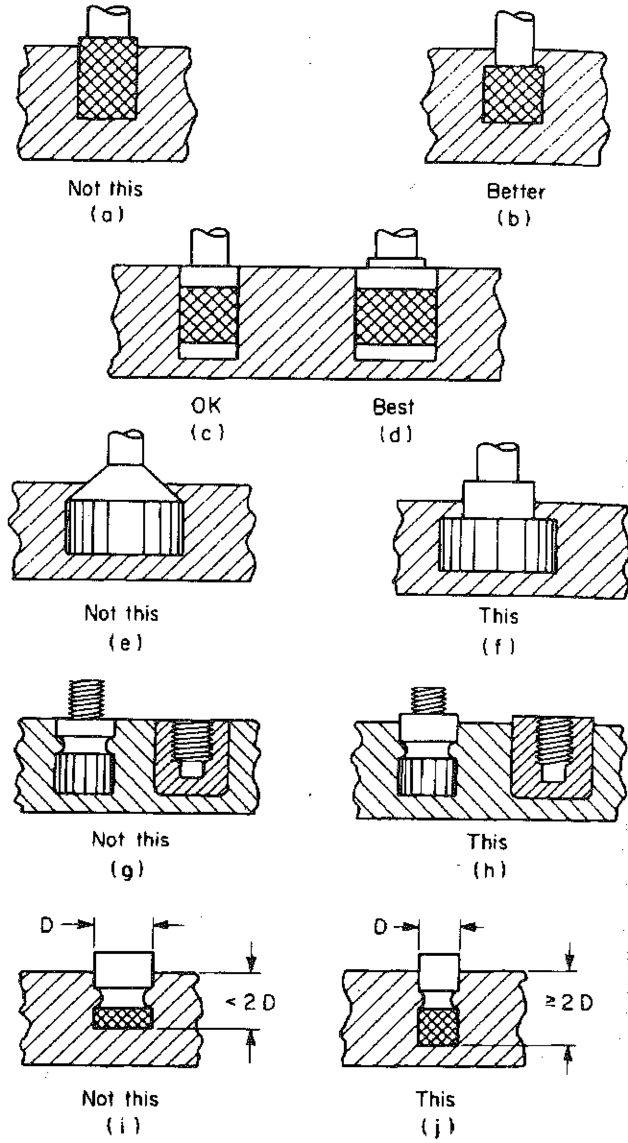

FIGURE 6.2.21 Recommended designs and placements for screw-machine parts to be inserted in plastic components. Design *a* is not a good design because there should be a smooth surface where the insert exits from the plastic. Design *b* is better, but it does not allow the flow of plastic material to be sealed off from unwanted areas. Design *c* is better from this standpoint. Design *d* is still better because it provides a double-sealing surface. Design *e* is not satisfactory because it would result in a featheredge of plastic material around the insert. Design *f* avoids this. In view *g*, the two parts are apt to suffer contamination of the screw threads with plastic material. Design *h* avoids this problem by raising the threaded portion of the inserts above the surface of the part. In view *i*, the part is not sufficiently embedded in the plastic material of the part. As shown in *j*, the depth of insertion should be at least 2 times the insert diameter.

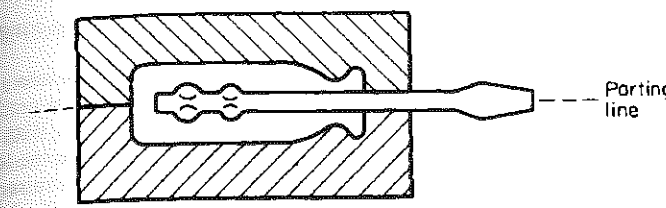

FIGURE 6.2.22 Irregularly shaped inserts are placed on the parting line of the mold.

Screw-machine inserts should be placed perpendicularly to the parting line of the mold to facilitate placement and avoid complications in mold construction. Irregular inserted parts such as screwdriver blades with noninserted widths or diameters larger than the inserted diameter are placed with their axes on the parting line of the mold. Otherwise, side cores must be provided to permit removal of the insert from the mold. (See Fig. 6.2.22.)

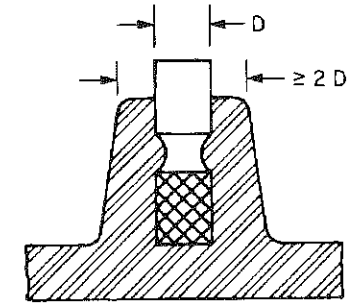

FIGURE 6.2.23 Ample supporting material must be provided around an insert. If the diameter of the insert is ¼ in or less, the boss diameter should be at least twice that of the insert.

Inserts very often are incorporated in a boss that provides supporting material for the insert. If the outside diameter of the insert is less than 6 mm (¼ in), the outside diameter of the boss should be twice that of the insert. (See Fig. 6.2.23.) If the outside diameter of the insert is larger than 6 mm (¼ in), wall thickness should be 50 to 100 percent of the insert diameter. Whenever the configuration permits, it is desirable to design the insert so that the flow of plastic is sealed off from threaded areas and other areas where plastic material is not intended to be. A good rule of thumb is to make the embedded length of an insert twice its diameter.

It is often advisable to press in the insert after molding. This avoids problems of contamination of the exposed surface of the insert with plastic material. It also avoids the possibility of damaging an injection mold with a misplaced insert. Ultrasonic techniques are particularly reliable for inserting metal inserts into plastics parts after molding. (See Chap. 6.9.)

Threaded portions of inserts should be raised from the surface of the molded part to avoid contamination of the threads with material. Also, featheredges of plastic material around inserts should be avoided.

Lettering and Surface Decorations

Lettering and other raised or depressed surface decorations and textures are easily incorporated into plastic parts. Once the lettering has been incorporated in the mold, each part will automatically show the lettering with few or no extra steps.

In most cases, mold cavities are machined rather than hubbed (pressed) or cast. With machined molds, the lettering in the *part* should be *raised,* i.e., formed by depressed, engraved letters in the mold. It is easier to engrave lettering in a mold cavity than it is to machine away the background and leave raised letters.

Hubbed (or hobbed) dies are made by pressing a hardened-steel form into annealed die steel. Very high pressures are required. This method is economical for producing a number of identical cavities, particularly of a difficult-to-machine shape. With hubbed dies it is best to have *depressed* letters in the part. This is true because the letters in the hub are depressed (engraved) and the letters in the mold are therefore raised.

With either raised or depressed letters, the letters should be perpendicular to the parting line of the mold. Otherwise, the part will have an undercut.

Sometimes it is desired to have depressed letters on the part and to fill them with paint that contrasts with the color of the plastic material. Cavities for filled lettering should be sharp-edged and 0.13 to 0.8 mm (0.005 to 0.030) in) wide. They should be one-half as deep as wide and should have a rounded bottom. (See Fig. 6.2.24.)

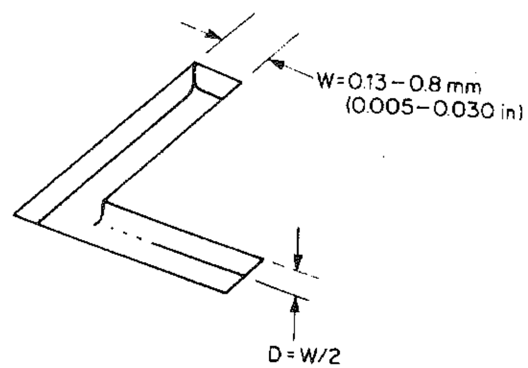

FIGURE 6.2.24 Dimensional rules for depressed lettering which is to be filled with paint.

Draft

It is highly desirable to incorporate some draft, or taper, in the sidewalls of injection-molded parts to facilitate removal of the part from the mold. Draft may not be necessary if ejector pins can be placed properly, but it still is wise, if the design permits, to make an easily removable part with draft and generous radii.

Drafts as low as ¼° are often adequate. Usually, deep parts require less draft angle than shallow parts. For shallow parts draft should average ½° or more; for deep parts ⅛° can often be satisfactory. Textured surfaces require greater draft.

The following are recommended minimum drafts for some common materials:

Polyethylene	¼°
Polystyrene	½°
Nylon	0–⅛°
Acetal	0–¼°
Acrylic	¼°

Corners: Radii and Fillets

Sharp corners should be avoided except at the parting line. They interfere with the smooth flow of material and create possibilities for turbulence with attendant surface defects. Sharp corners also cause stress concentrations in the part that are undesirable from a functional standpoint.

Fillets and radii should be as generous as possible. A desirable minimum under any circumstance is 0.5 mm (0.020 in), while 1.0 mm (0.040 in) is a preferable minimum if part requirements permit.

Surface Finish

One significant advantage of the injection-molding process is the fact that surface polish or textures can be molded into the part. No secondary surface-finishing operations (except, of course, plating, hot stamping, or painting, if desired) are necessary.

High-gloss finishes are feasible if the mold is highly polished and if molding conditions are correct. However, dull, matte, or textured finishes are preferred to glossy finishes, which tend to accentuate sink marks and other surface imperfections.

Painting of most thermoplastics is feasible but is not recommended if the color can be molded into the part. The latter approach obviously is more economical and gives superior results. If contrasting colors are required, masks can be fabricated and a portion of the part left unpainted. See Chaps. 6.12 and 8.5 for further information.

Plating of plastic materials is feasible for some plastics but is a specialized operation. See Chap. 8.2 for more information.

Surface decorations such as flutes, reeds, and textures should stop short of the parting line so that any parting-line flash is easy to remove. (See Fig. 6.2.25.)

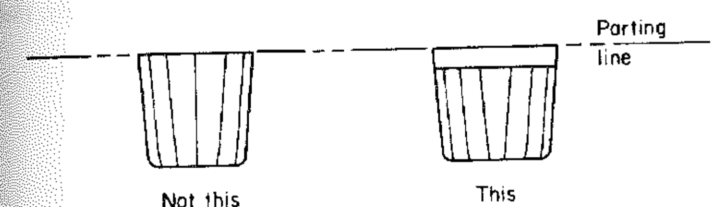

FIGURE 6.2.25 Surface decorations like flutes, reeds, and textures should stop short of the parting line so that parting-line flash is easy to remove.

Flat Surfaces

Flat surfaces, although feasible, are somewhat more prone to show irregularities than gently curved surfaces. Since the latter also produce more rigid parts, they are preferable.

Mold Parting Line

Every injection-molded part shows the effect of the mold parting line, the junction of the two halves of the mold. The part and the mold should be designed so that the part-

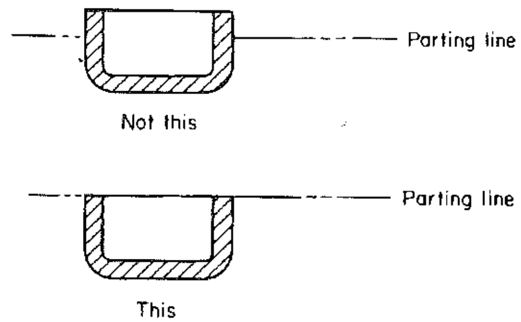

FIGURE 6.2.26 If possible, put the mold-parting line at the edge of the part.

ing occurs in an area where it does not adversely affect the appearance or function of the part. One easy way to do this is to put the parting line at the edge of the part where there is already a sharp corner. (See Fig. 6.2.26.) However, removal of parting-line flashing may destroy the sharpness of the corner. The part drawing specification should permit this.

Parting lines should be straight; i.e., the two mold halves should meet in one plane only. This obviously provides more economical mold construction, but it may not be possible if the part design is irregular.

If it is not possible to put the parting line at the edge of the part, cleaning parting-line flash is facilitated by having a bead or other raised surface at the parting line as shown in Fig. 6.2.27. Deliberately offset cavities are helpful in avoiding appearance defects, which may occur if the two mold halves do not line up properly. (See Fig. 6.2.28.)

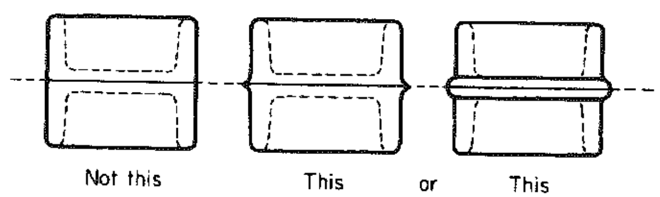

FIGURE 6.2.27 A bead at the parting line facilitates removal of mold flash.

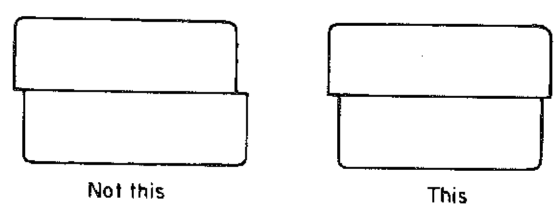

FIGURE 6.2.28 Deliberately offset sidewalls help avoid appearance defects if mold halves do not line up properly.

Computer Injection-Molding Simulations

These can be invaluable to both the product designer and tool designer. They aid in optimizing the material flow in the mold and, in the process, help avoid or minimize molding problems such as shrinkage, warpage, weld lines, and sink marks. Changes in both the mold and the part may result from such a simulation. Problems can be identified before the mold is actually machined. The need for prototypes is reduced. Software for such simulations is available from a number of sources.

Support Design for Assembly

Designers should endeavor to incorporate several functions into plastic parts so as to minimize the total number of parts to be assembled. By incorporating hinge, spring, fastener, and other functions in the injected-mold part, its cost may be only slightly increased, but separate parts for these functions will not be needed. The costs of designing, manufacturing, and stocking separate parts will be eliminated, and the assembly operation will be greatly simplified. Figure 6.2.29 illustrates one common step of this kind, the consolidation of a snap-fit element, which provides for easy assembly and eliminates the need for separate fasteners.

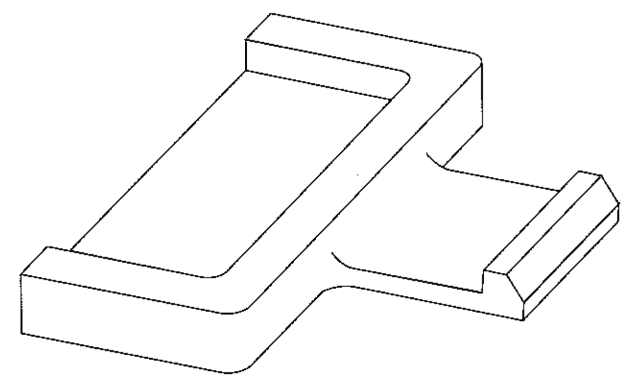

FIGURE 6.2.29 Snap-fit cantilever beams are often used in injection-molded parts to facilitate assembly.

DIMENSIONAL FACTORS AND TOLERANCE RECOMMENDATIONS

Though surprisingly tight tolerances can be held when molding thermoplastics parts, dimensions cannot be held with the precision obtainable with close-tolerance machined metal parts. There are several reasons for this:

1. There is materials shrinkage, as discussed above, including variation and unpredictability in the shrinkage.
2. Plastics exhibit a high thermal coefficient of expansion. As a result, if the toler-

ances are extreme, designers should specify the temperature at which the measurements should be taken.

3. Despite automatic-control apparatus for pressure, temperature, and time settings, there is some variation in these factors from cycle to cycle. These variations result in slight dimensional variations in molded parts.
4. Mold runners, cooling channels, and gates cannot always be located in the optimal position, leading to differences in how uniformly the material is "packed" in the mold and how uniformly it cools. Some distortion or built-in stresses are unavoidable.
5. Plastic parts are usually more flexible than metals.

A corollary of the flexibility factor is a lessened need for very close tolerances. Plastic parts, when assembled, often can be deformed slightly if this is necessary to ensure a good fit. Knowledgeable designers take advantage of this fact by designing lips and locating bosses on plastic parts to ensure alignment with mating-part surfaces when necessary. (See Fig. 6.2.30.)

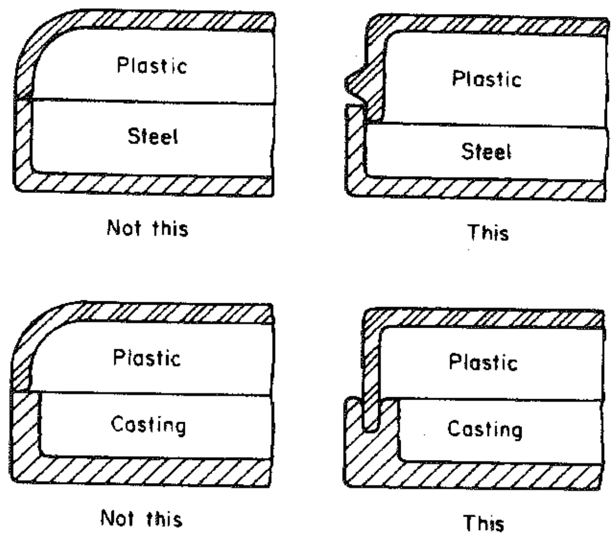

FIGURE 6.2.30 Lips and locating bosses aid in alignment when plastic parts are assembled.

As with other processes, close dimensional tolerances can greatly increase the cost of injection-molded parts. Fine-tolerance molds are more costly than looser-tolerance molds. There are processing-cost increases as well when extra tight dimensional control is needed. For example, closer process controls are needed for pressure, temperature, and cycle time; cycle time may be increased; shrink fixtures may be required after the part has been removed from the mold; and scrap rates will be higher.

Different plastics materials have different tolerance capabilities. Low-shrinkage materials can invariably be molded with closer tolerances. Glass-or mineral-filled materials can be molded more accurately than unfilled materials.

The use of a greater number of mold cavities tends to reduce the closeness of dimensional control over the molded parts. As a rule of thumb, for each cavity after the first, allowable dimensional tolerances should be increased by 5 percent. For example, a single-cavity mold with an allowable tolerance of ± 0.1 mm (0.004 in) on a particular dimension should have ± 0.15 mm (0.006 in) if the number of cavities is 10 (10×5 percent = 50 percent increase in tolerance).

Tables 6.2.4 and 6.2.5 show suggested values for dimensional tolerances for various plastic materials. These tables, developed from data supplied by the Society of the Plastics Industry, represent historic and customary practices prevailing in the plastics-molding industry. Contract forms or other agreements of individual molders may vary.