HANSER

Sample Pages

Jordan Rotheiser
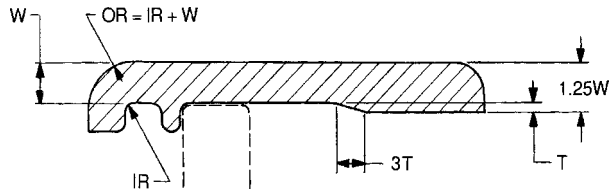
Joining of Plastics

Handbook for Designers and Engineers

For further information and order see

Figure 2-2   Improved design

This statement applies principally to thermoplastics, of which those with high post-molding shrinkage rates are somewhat more vulnerable to this condition than the others. Fillers will reduce the effect of this phenomenon much as they affect shrinkage. Structural foam parts will not experience this condition, and it is significantly reduced in thermosets.

Even when compliance with this rule is maintained, the change in wall thickness should be gradual as shown in Fig. 2-2. When the mold is filled, the flow of material should be from thick section to thin section.

Presumably, the thick wall on the left side of Fig. 2-1 was initially dictated by the need to locate a component. That component is illustrated by the dotted line in Fig. 2-2, where the thick corner has been modified to a uniform wall and a locating rib. The thick wall section has been reduced to the prescribed maximum. Thus the function has been maintained and the source of the nonuniform cooling condition has been eliminated.
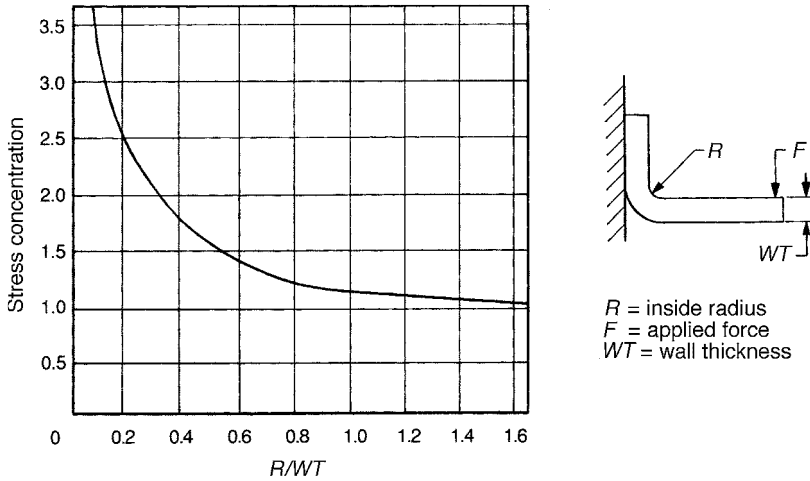
2.2   Inside Corner Stress

Elimination of internal stress due to nonuniform cooling does not remove all the stresses resulting from the part contour. Sharp inside radii are another source of such stress. Figure 2-3a illustrates the stress pattern resulting from a sharp corner. Figure 2-3b shows how the stress is distributed in a rounded corner. This effect can be seen when a clear part is observed under polarized light.

The effect of corner radius on stress concentration is clearly illustrated by the curve plotted in Fig. 2-4. Note that the curve rises sharply at the 0.25 level. Therefore, an R/WT ratio of 0.25 should be regarded as a minimum. Note, also, that the curve flattens out beyond the 0.8 ratio. *A ratio of 0.5 should be regarded as optimum.* If the two walls are not equal, the thinner wall may be used for this computation. Reinforced materials are better able to withstand stress than unreinforced plastics. In that case, the curve of Fig. 2-4 can be disregarded to some extent. The inside radius, however, should never be less than 0.50 mm (0.020 in.).
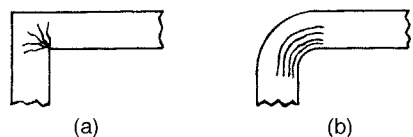
Figure 2-3   Stress patterns: (a) corner stress concentration and (b) relieved stress

Figure 2-4   Relationship of inside corner radius to stress concentration factor


(a)   (b)
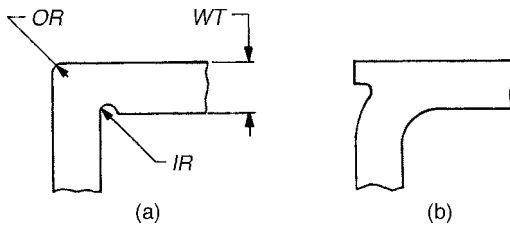
Figure 2-5   Sharp corner designs: (a) inside coring and (b) "drawer front" configuration

Aesthetic requirements sometimes require sharp outside corners. In those cases, the designs illustrated in Fig. 2-5 can be used to approximate this appearance.

2.3   Ribs and Bosses

The use of longitudinal and box ribbing significantly increases the strength of the part. A method for the application of the basic design rules to the design of ribs is illustrated in Fig. 2-6, where a circle has been drawn at the intersection of the rib and the wall. The circle has three points of contact: at the nominal wall and at each inside radius at the base of the rib. The diameter (D) of this circle must not exceed 1.25 times the nominal wall (W). The same device may be used for the intersection of two ribs, adjacent bosses, a rib and a boss, etc. Care must be taken to identify the thickest condition correctly. For example, in the case of the intersection of two ribs, the circle must be drawn through the diagonal radii.

In the design of ribs or bosses, whose bases must meet the same criteria, competing factors must be considered. Increasing the inside radius to reduce the stress concentration factor results in less material thickness available for the rib. Figure 2-6a
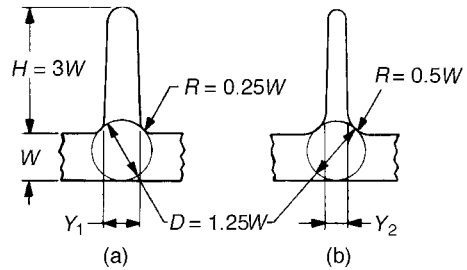
Figure 2-6   The 25% rule for ribs:
(a) $R = 0.25W$ and (b) $R = 0.5W$

illustrates a rib constructed with an inside radius of $0.25W$. Figure 2-4 indicates that the stress concentration factor for that inside radius is about 2.3. To reduce the stress concentration factor to 1.5, the inside radius would need to be increased to $0.5W$ (Fig. 2-6b). That results in a wall thickness at the base (Y_2) of the rib of $0.5W$, a significant reduction from the original thickness (Y_1). Obviously the rib wall thickness will provide less strength with less material. However, the overall strength provided by the rib may be greater because the internal stress has been reduced. If more strength is required, additional ribs are a better solution than a rib with high levels of molded-in stress.

   There is an additional consideration in the design of ribs and bosses: the thinning down of the rib due to draft. Not only does this result in further thinning of the rib, it could result in a rib thickness at the top that is too thin to fill or one that is so thin it can be filled only at elevated temperature and pressure. In short, such thinning could be the driving force in determining the molding conditions, the nature of which increases the likelihood of distorted parts. For example, suppose that the value of W in Fig. 2-6b is placed at 2.5 mm (0.100 in). Then radius r would be 1.25 mm (0.050 in.), resulting in a rib thickness of 1.25 mm (0.050 in.) at its base. The recommended maximum rib height is three times the nominal wall thickness (more will not significantly increase strength). For a rib height H of 7.5 mm (0.300 in.) and a recommended draft of 2°/side, the rib would have a thickness of 0.86 mm (0.34 in.). That would be difficult to fill in the mold with a viscous material, particularly if the rib were located at a point distant from the gate and with a variety of contours in the flow path, which would cause pressure drops.

2.4   Draft

The part shrinks down on the core as it cools. Draft is used to help the part break free of that core. Although the temptation may be present, *do not give serious consideration to the elimination of draft.* If the draft angle is reduced, the locking force becomes greater, increasing the amount of pressure required to remove the part from the tool. The molding cycle will need to be extended to allow the part to cool enough to be ejected. The part must cool until it is rigid enough to withstand the pressure of ejection without distortion. This can be 50 to 80% of the molding cycle. The greater the draft, the lower the force necessary to eject the part, and the sooner ejection is possible.
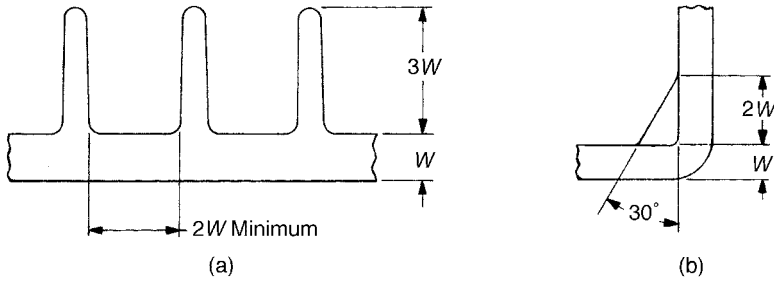
Figure 2-7   Ribs and gussets: (a) multiple ribbing and (b) gussets

A reduction in draft angle to one degree is a reasonable compromise between these competing demands for most materials. Unfortunately, it is too easy to allow one degree to become a habit, thus ignoring the price it is extracting.

The locking effect described above is increased with ribs because they provide additional gripping surfaces. Multiple ribs are worse; however, they are preferable to a rib without draft or a thick one with high levels of molded-in stress. Figure 2-7a demonstrates the minimum spacing between ribs of twice the nominal wall thickness. It is always desirable to attach ribs to a wall. This permits the gas (air) that is in the mold to escape the rib cavity recess. Freestanding ribs can result in gas traps.

Figure 2-7b illustrates the recommended configuration for gussets. A height greater than $2W$ will not significantly increase the strength. The same spacing rule which applies to ribs ($2 \times W$) would apply for gussets.

A general guideline of 2°/side is desirable for outside walls as well. The effect of draft is demonstrated in Fig. 2-8. Note the flat angle in section A. Obviously it would have no difficulty separating from core or cavity.

Section B illustrates a 2°/side draft angle. It will separate from the mold, but considerably greater force will be necessary. However, compared to section C, which has no draft, it will be readily ejected. Without draft, section C will require tremendous force to overcome the effects of shrinkage on the core. The part may be too distorted to assemble in addition to needing more cycle time. Also there will be drag marks on the part where there is inadequate draft.

Draft must be present on all surfaces perpendicular to the parting line.
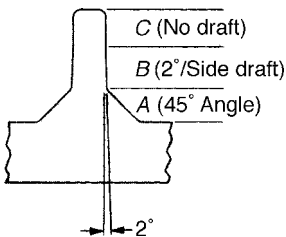
Figure 2-8   Effects of draft

There is, however, one exception to this rule with regard to the leading edge of a locating rib. This is described in Section 2.7.1.

It should be noted that the 2°/side direction represents an increase from the angle recommended for many years, namely, 1°/side. Several reasons account for this change. First, molding cycles have decreased, and there is strong economic pressure to eject parts faster than previously. Second, consumer expectations have been elevated by competition. Parts with slight scuff or drag marks from too little draft on the tool are no longer acceptable. Additional draft is preferable to painting the part to achieve an acceptable finish, an alternative imposed by competition in some cases. Finally, the combining of parts in the ongoing effort to eliminate assembly operations has resulted in components of greater complexity. Such parts have more detail, which increases the gripping effect of the part in the mold.

When texture is applied to the outside wall of a part, it results in tiny undercuts in the surface. To accommodate this surface, additional draft must be provided. While the draft allowance can vary from pattern to pattern and from one engraver to another, the most generally accepted rule is:

> **Allow 1.5°/side plus 1.5° for each 0.001 in. depth of engraving.**

Thus, for a texture depth of 0.025 mm (0.003 in.), allow a draft of 6°/side. The draft allowance should be approved by the engraver. When the amount of draft so determined interferes with critical requirements, the engraver may be able to feather the engraving to some extent. On some cases, less draft is acceptable. Regardless of the number of shops that manufacture molds for the product, all the engraving should be performed by the same engraver (in the same location), to ensure the greatest uniformity of engraving.

2.5   Shrinkage

Basic physics describes the phenomena whereby objects expand with heat and contract upon cooling. An object not constrained in some fashion behaves in exactly that way. When a plastic part cools in a mold, however, it is constrained by the mold. Usually, the core of the mold constrains the part, although such part details as external ribs can result in the cavity constraining the part.

The molten plastic begins to cool the instant it enters the mold. When the mold opens and the cavity side is removed, the part is exposed to the air and cooling commences in earnest. As the part cools, it stiffens. When it has done so enough to withstand the stress of ejection, it is removed from the mold.

At this point, the part is no longer constrained. This is the stage at which most of the shrinkage takes place, hence the term "post molding shrinkage." The majority of this shrinkage (75–95%) occurs within the first 2 h. By the end of a week, nearly all of it has taken place. Crystalline thermoplastics, however, can take a year to reach final equilibrium.
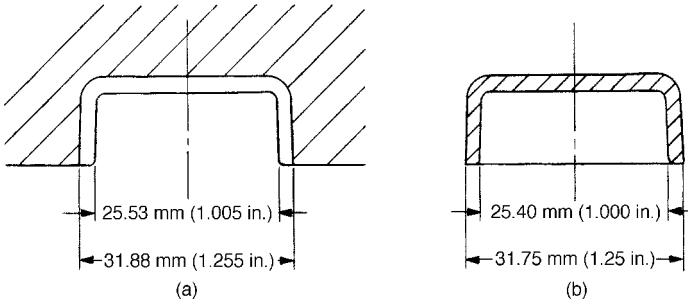
Figure 2-9   Postmolding shrinkage: (a) core and cavity and (b) finished part

This phenomenon must be considered when post molding operations such as machining or assembly are planned. As a general rule, it is wise to wait 24 h before performing any machining operations more precise then edge or gate trimming. The same guideline applies to assembly operations. However, in some cases, it can be an advantage to assemble two parts while one is still hot, provided the hot part is the female fitment. Cooling will then result in a firm joint as the outside part cools and shrinks down on the inside part.

Postmolding shrinkage figures for resins are readily available from the manufacturer. Generally, the moldmaker will add an allowance for shrinkage to the dimension. That is the number to which the mold core and cavity are built. Figure 2-9a illustrates the dimensions to which the mold for the part shown in Fig. 2-9b would be built. Those numbers were arrived at in this fashion.

$$(1 + a)b = c \tag{2-1}$$

where a = shrinkage rate

b = part dimension

c = mold dimension

A multiplier is created by adding the shrinkage rate to the number. Thus, a shrinkage rate of 0.005 in. per inch (a) would result in a multiplier of 1.005. Therefore: for a part dimension of 25.4 mm (1.000 in.), the mold dimension would be

1.005×25.4 mm $= 25.53$ mm

or

1.005×1.000 in. $= 1.005$ in.

Shrinkage rates are often indicated as a range. Usually the lower figure applies to thin walls and the higher figure to thicker walls. Shrinkage rates for amorphous thermoplastics tend to be low; however, those for crystalline materials can be large and can have an enormous range. Some polyethylene resins have shrinkage rates ranging from 0.38 mm/mm (0.015 in./in.) to 1.27 mm/mm. (0.050 in./in.)

Shrinkage rates have a direct relationship to assembly considerations with respect to material selection and joint design. Once the parts have been designed and the mold constructed, engineers' materials selection options are limited to resins with like shrink

age factors. Thus, it would be impossible to switch from a polyethylene with a shrinkage of 0.75 mm/mm (0.030 in./in.) to a polystyrene with a shrinkage of 0.10 mm/mm (0.004/5 in./in.). The part dimensions would be substantially different.

Of course, this can be used to advantage. If one of the parts is a little large and the mold is difficult or impossible to alter, a version of the material with a slightly higher shrinkage could be used. Conversely, a material with a slightly lower shrinkage or with a filler added could solve the problem if the parts were too small. Molding conditions can also have a significant affect on shrinkage.

2.6   Fitments

2.6.1   Drawing Conventions for Plastic Assembly

It is important for designers and engineers to understand the legal aspects of engineering drawings. A contract in the form of a purchase order that is let for the tooling for the new part generally refers to the engineering drawing as the source for the dimensions and tolerances to which the tooling will be made. Thus, that drawing becomes part of the contract, and the toolmaker is obligated to incorporate every detail described on the drawing within the tolerance limits provided. The toolmaker cannot legally protest that he is entitled to additional money because of details or tolerances he failed to notice when he quoted the mold. Conversely, he has the right to demand additional fees and production time for details or tolerances the designer or engineer failed to include in the drawing when it was originally released for tooling.

It therefore behooves all involved in the design and development process to pay engineering drawings the respect due a legal document.

Efforts to reduce the amount of time required to bring a new product to market have brought great pressures to bear on the design community. Unfortunately, in some cases this has resulted in drawing shortcuts that have led to the production of parts that do not fit together. The consequence of this circumstance is usually mold revisions, which are both costly and time consuming. In some cases, the mold cannot be salvaged and a complete new one must be built. Legal disputes may ensue.

One of the most serious of such ill-advised shortcuts is the failure to draw draft angles and corner radii. Both these fundamental features of plastic parts have a significant effect on their assembly and on the cost of their tooling. Designers and engineers from other disciplines, however, tend to underestimate the importance of these basics. The problem is exacerbated when the drawing is inconvenient for the moldmaker to check because it contains few or no dimensions. When a three-dimensional computer-aided design program is used, the file will be transferred directly to the machining center and the mold will be built exactly as the object is drawn.

Figure 2-10a illustrates the type of drawing just referred to. If such a drawing contains notes that state "Allowable draft 1°/side" and "1.5 mm (0.059 in.) inside radius permitted," it is a recipe for disaster. The use of the words "allowable" and "permitted" gives the moldmaker every right to build a mold that will make a part just as shown in the drawing. In fact, it would be less costly for the moldmaker to make the
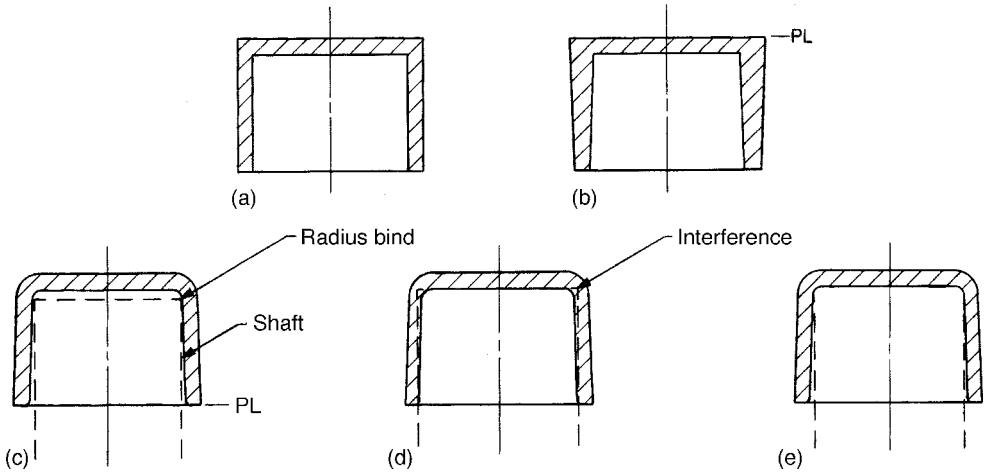
Figure 2-10   Drawing effects: (a) part drawn without radii or draft, (b) part made with parting line at top, (c) part made with parting line at bottom, (d) potential for interference based on mating part without draft consideration, and (e) fitment with draft considered

mold exactly that way. It costs money to add radii and draft walls. For that matter, the moldmaker could easily have made it as shown in Fig. 2-10b with the parting line at the top. The drawing provides that freedom. Best intentions notwithstanding, anything that is not on the drawing is not in the contract. Molds have actually been made just as shown in these drawings.

A prudent policy to eliminate any possible misunderstandings is to follow this rule:

The drawing must represent the part exactly as its shape is intended.

Thus, the part that was depicted in Fig. 2-10a should actually be represented as it is shown in Fig. 2-10c. Both draft and radii are drawn into the part, clearly showing some of the potential fitment problems. The dashed line demonstrates the interference that would occur if the mating part were made with a square corner.

The dashed lines in Fig. 2-10d demonstrate the interference that would occur if the mating part were made without consideration for draft. If the stated dimension were taken as the maximum number in the draft, the fitment would have turned out as in Fig. 2-10d. If the stated dimension were taken as the minimum number in the draft, the fitment would have turned out as in Fig. 2-10e.

There are so many computer-aided design (CAD) programs in general use that it simply is not economically feasible for moldmakers to maintain seats for them all. Therefore, each selects the one or two best suited to his own operation. Drawings provided in other programs are converted to the one at hand. When a CAD file is provided that can be used directly in the moldmaker's system, much of the responsibility for the accuracy of the mold shifts from the moldmaker to the CAD

designer and his company. In such cases the CAD designer must exhibit great care lest walls without draft and radiused corners appear in the mold. He or she must also be aware that two levels of design inspection have been removed from the system: the checker and the moldmaker.

2.6.2   Importance of Tolerancing for Assembly

If football can be called a "game of inches," then assembly must be called a "game of thousandths of inches." The difference between a force fit, solvent weld, or other type of joint that provides adequate strength and one that fails can literally be just a very few thousandths of an inch or hundredths of a millimeter.

A vast fortune in extra mold costs and mold revisions has resulted from improper tolerancing. In some cases, projects have been canceled because of a failure in joining that could be directly traced to poorly executed tolerances. In other cases, large multi-cavity molds have been routinely operated at partial capacity because all the cavities could not be kept functioning within tolerance.

Many of these problems can be solved with better dimensioning practice. Once the object is drawn as it is intended to appear, it becomes apparent that careful control of fitment details is critical to the success of the joining.

Dimension A in Fig. 2-11 is one that does not actually exist either in the mold or in the part. While it can be measured in the mold before the radius is added to the core, it is useful only as a reference dimension because it does not exist in the part. The first point that can be measured in either the finished mold or the part is the tangent to the radius, dimension B. This distance can be significantly different from dimension A. For a draft angle of 2°/side and an outside radius of 3.18 mm (0.125 in.), dimension B would be 0.23 mm (0.009 in.) greater than dimension A. That is sufficient to invalidate a tolerancing scheme. Furthermore, it could cause a close fitting part to be incorrectly deemed in or out of tolerance by quality assurance personnel.
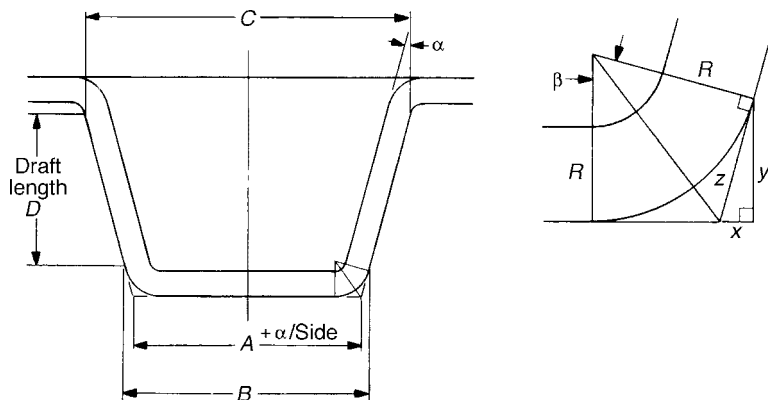

Figure 2-11   Fitment tolerancing

CAD programs will provide the dimension at the tangent. For those who wish to determine this location manually, the computations are as follows:

$$\angle\beta = 90° - \angle\alpha \tag{2-2}$$

$$z = R \tan \tfrac{1}{2}\beta \tag{2-3}$$

$$x = z \sin \alpha \tag{2-4}$$

$$y = z \cos \alpha \tag{2-5}$$

where $\angle\alpha$ = draft angle

$\angle\beta$ = construction angle

z = construction hypotenuse

x = horizontal offset

y = vertical distance to tangent

One of the problems encountered in the practical application of this principle is the tendency of moldmakers to presume that the dimension is meant to be to the intersect even when it is indicated to the tangent. This is a result of the widespread practice of dimensioning the intersect instead of the tangent. Thus it behooves the designer to show very clearly where the dimension is taken to, even to the point of providing both dimensions.

The dimension at the other end of the part, dimension *C*, is of equal importance and the same condition with regard to a square corner would also apply. Its location is determined by Eq. 2-6. The generally accepted drafting rules state that if a dimension and the related angle are toleranced, the dimension at the other end must be referenced. However, if the fitment requires close control, it may be better to tolerance the dimensions at each end and permit the angle to be referenced.

$$x' = D \tan \alpha \tag{2-6}$$

where x' = horizontal offset of wall

α = draft angle

D = length of the drafted wall

This practice prevents the problems that result when a toleranced end is carefully controlled and the designer fails to check the range of dimensions at the other end resulting from the angle tolerance. For example, for an angle tolerance of $\pm1/2°$ and a height *D* of 25.4 mm (1 in.), dimension *C* could vary by 0.43 mm (0.017 in.) easily enough to get a fitment in trouble.

2.6.3   Special Drafting Practices for Plastics

One drafting practice that is somewhat unique to the plastics industry is the designation for draft. It is usually specified as $+\alpha°$/side and placed on the dimension taken from that point. Thus, the designation $+2°$/side indicates that a 2° draft is

intended to increase from the point of the dimension so indicated. Conversely, the designation –2°/side would indicate that the draft decreases from the point dimensioned by 2°/side.

In addition, several notes commonly used on plastics drawings are not typically found in drawings for other industries.

1. *Each cavity must contain an identification number – location to be approved.* This refers to the common practice of using multiple cavities, which will vary in dimension and should be identified by a small number or letter located where it will not interfere with a fitment or the appearance of the part. Identification is necessary to locate a cavity that is not producing acceptable parts.

2. *Gate location must be approved.* A number of plastics processes require a gate through which the mold is filled. Neither the molder nor the moldmaker has a thorough understanding of the function of the part or its mating fitments. Therefore, the designer or engineer must approve the location of the gate to ensure that it does not interfere with the function or the appearance of the part. That statement notwithstanding, the processor should be given all the freedom possible to permit the production of the part in the most economical manner.

3. *Weld line must not be visible.* Parts from gated processes will contain a weld line (knit line) that will be located, depending on the contour of the part, at a point approximately opposite the gate. Multiple gates will result in multiple weld lines. There will also be a weld line around each core pin, since the material must flow around that pin to fill the mold. Unless the hole is drilled, a hole and a boss will always have a core pin and, therefore, a weld line. Cored holes should not be positioned less than 1.5 to 2.0 mm (0.059 to 0.079 in.) from the edge of an injection molded part; or two diameters of the largest cored hole from the edge of a compression molded part.

   The temperature and pressure of the melt will determine the amount of weld achieved at the weld line. "Open weld line" is the term used to indicate that no weld at all has been achieved. Such a condition always results in a reject part. In some cases, a partial weld is achieved. This will result in a visible weld line much weaker than the surrounding material. As indicated above, this is also a reject condition. A good weld line will not be noticeable, although a slight line may be visible on a polished surface if closely inspected. This can be acceptable; however, testing of weld lines to establish acceptable limits may be necessary. This may require the words "limit samples" to be added to note 3. Even a good weld, however, is somewhat weaker than the surrounding material, and that is where failure is most likely to occur. Critical areas that cannot withstand any loss of strength will need to be flagged with the notation "No weld line permitted in this area" so the gating can be located accordingly. Methods of dealing with bosses in regard to boss cracking due to self-tapping screws are discussed in Chapter 8, Fasteners and Inserts.

4. *Maximum allowable flash is .XXX in.* Flash is the tiny amount of plastic that fills the crevice between the mold components. It is almost always present in a part molded under optimum conditions. The amount varies according to the process. To demand no flash brands the designer as a neophyte and may lead to costly

post-molding trimming operations, which are truly unnecessary. A far better practice is to design the part so the fitments do not occur on the parting line. Failing that, the next best approach is to design the fitment to allow for a reasonable amount of flash, which will avoid an increase in the cost of the part.

5. *Maximum allowable mismatch is .XXX in.* "Mismatch" generally refers to the alignment between the upper and lower parts of the mold at the parting line. However, it can be construed to refer to the alignment between any two mold components anywhere on the core or cavity. To avoid disputes over whether any misalignment is or is not included in the tolerance, maximum allowable mismatch should be so indicated. Misalignment between core and cavity results in a variation in the wall thickness between the two sides. Furthermore, since the cavity space is different between the two sides of the mold, the flow of the molten plastic in the mold will be altered, potentially affecting the strength and location of a weld line at a hole or boss. Misalignment between core and cavity is also controlled by placing a tolerance on the wall thickness.

6. *Part must be flat within .XXX mm (or within .XXX mm/mm).* It has been said that there is no such thing as an absolutely flat or perfectly round plastic part. Perhaps not, but it is certainly within reason to set limits on how much out of flat or out of round a part can be. Plastics processes are cycle sensitive. Without such limits, the process can go out of control and proper fitments can no longer be guaranteed.

7. *Material is to be [name of manufacturer]   [exact number of resin]. Part is to include XX% additive [name of manufacturer]   [exact number of additive]. No substitutions permitted without written authorization.* The material, which has been tested and approved, should be clearly indicated and no substitutions permitted. That is because there is no such thing as a competing material exactly the same as the original material. Patents prevent that. Thus, another material will behave differently in some respects and will have somewhat different properties. Depending on the application, such variations may or may not impact the performance of the product, with potential legal implications.

    Also, the material supplier may not be the actual resin producer. In fact, the supplier may use several sources. His practice must be ascertained, and resin from all possible sources tested for critical applications.

    In his exercise of due diligence, the engineer will test additional resin grades until at least two can be approved (three is better). This provides the molder with some flexibility to respond to availability and pricing fluctuations thus keeping costs under control. The vague term "or equivalent" should be avoided at all times, since it is subject to interpretation, which in turn can result in disputes.

8. *XX% regrind acceptable.* "Regrind" is the term used to refer to sprues, runners, and rejected parts that have been reground so they can be mixed in with the virgin material and run through the molding machine again. The use of regrind is one of the characteristics of thermoplastics that has made them economically competitive by enabling nearly 100% material utilization and solves the problem of in-process recycling. However, the material suffers some degradation when it is raised to elevated temperatures. Thus, regrind reduces the physical properties of

the material. However, the degree of drop varies between materials and according to molding conditions.

Obviously, the higher the amount of regrind permitted, the greater the drop in physical properties. However, this diminution is magnified by the multiplier effect which takes place. For example, if 20% regrind is used, the batch will contain 4% (20% of 20%) that has been through twice, 0.8% that has been through three times, 0.16% that has been through four times, etc. Readily visible signs of degradation are an increase in brittleness and a yellowing or darkening of color.

9. *Ejector locations must be approved.* The designer must bear in mind that neither the molder nor the moldmaker is intimately familiar with the product or its application. These suppliers will select ejector pin locations based on mold construction and part ejection criteria, and their choices may conflict with fitment or other features in the part. Therefore, the locations selected must be approved by the designer.

Be particularly alert to the location of ejector pins on ribs. Ribs are a logical place for ejector pins because of their gripping effect on the core of the mold. However, often the rib is too thin to support any but the smallest diameter ejector pins, the type that break often and need to be replaced. The moldmaker can solve this problem by using a larger diameter pin. The larger diameter pin will require a circle of material to push on. This is known as an ejector pin pad and is illustrated in Fig. 2-12. If made too large, however, it can lead to a sink on the outside surface. (*Ref. 2.3, Ribs and Bosses*)

Before agreeing to this location, the designer should check that the pin pad or recess will not interfere with any fitments nor create an unacceptable sink mark at the intersection with the outside wall.
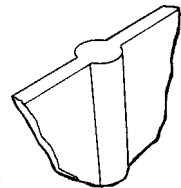
Figure 2-12   Ejector pin pad

10. *SPI Finish No. X on all outside surfaces except as noted. SPI Finish No. Y on all inside surfaces.* The pattern, location, and depth of finish should be clearly indicated across the area of intended application. Exterior surfaces must meet appearance requirements, which may include sections that vary from the widely used SPI finishes. Interior finishes must be polished just enough to avoid the need to use increased force for ejection due to rough, unpolished interior surfaces, which create tiny undercuts that cause the part to adhere to the core.

11. *Tolerances ±.XXX if not otherwise specified.* When properly employed, this note, or a tolerance box, is a convenience that can save considerable drafting time. Unfortunately, more often than not, it is not properly employed and it leads to excessively tight tolerances. That is because it is easier for the designer to use the overall tolerance than to work out the tolerance for each location. When the range

of tolerances used is great, it may be more useful to use the note "Dimensions reference if not toleranced." ("Reference" means for informational purposes only – usually abbreviated REF.)

Excessively tight tolerances increase mold and molding costs considerably. The moldmaker must build the tool to tolerances one-third those indicated on the drawing, the balance being reserved for molding variations. A common blanket tolerance is ±0.13 mm (±0.005 in.). That means the moldmaker is building those dimensions to a tolerance of ±0.043 mm (±0.0017 in.). That is careful, precision work. It also means slow, expensive work, which often unnecessarily lengthens the amount of time required to build the tool. For his part, the molder must slow the molding cycle to meet tight tolerances. Clearly, a large portion of the cost of a part is in the tolerancing.

Unfortunately, casual requests for ultra-tight tolerances are so widespread that moldmakers and molders have become accustomed to asking engineers to indicate which tolerances they really want held. The establishment of so-called critical dimensions undermines the validity of the entire tolerancing system. All tolerances should be held, and none should be asked for that are not necessary. All parties to the transaction should bear in mind that the drawing is part of a contract. In the event of a disaster that leads to legal action, what is written on that drawing is what will count the most.

The foregoing reminder of the legal standing of engineering drawings is not to be interpreted to mean that a deviation from the drawing tolerance cannot be approved. Tolerances created on paper before the part is molded sometimes turn out to be excessively tight when the actual part is available. This is particularly true because there is no inexpensive method of predetermining the actual rigidity of the part in advance of manufacture. Therefore, the actual part may flex more than anticipated, and greater tolerances may be acceptable. When parts are accepted with deviations from the contract drawing, a written record should be retained and the drawing should be altered to reflect the newly approved tolerance.

Regardless of how specified, the objective remains the same: the parts must fit together readily and stay together within acceptable parameters.

2.6.4　　Procedure for Establishing Tolerances

Figure 2-13a illustrates a typical male/female fitment. Its tolerances will be developed as in Fig. 2-13b. Presuming a desired dimension of 25.40 mm (1 in.) for the male fitment, the trial tolerance of ±0.13 mm (±0.005 in.) is applied. That will result in a high side of 25.53 mm (1.005 in.) and a low side of 25.27 mm (0.995 in.). Since the starting point is the male fitment, the high side will establish the minimum clearance. If a clearance of 0.05 mm (0.002 in.) is determined to be the most desirable, the lowest acceptable dimension for the female fitment becomes 25.58 mm (1.007 in.). If the same tolerance of ±0.13 mm (±0.005 in.) is retained, the nominal dimension for the female fitment is 25.70 mm (1.012 in.) and the high side becomes 25.83 mm (1.017 in.).
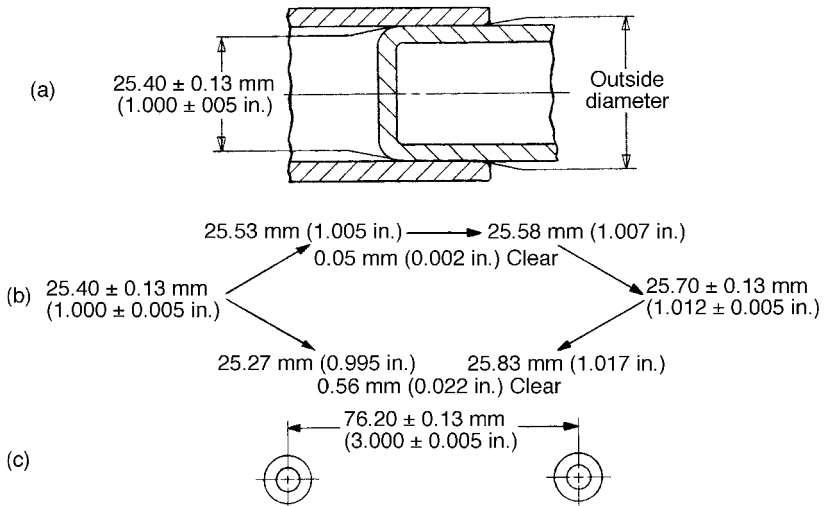
Figure 2-13   Tolerancing procedure

Referring to the lowest acceptable dimension for the male fitment of 25.27 mm (0.995 in.), the maximum clearance becomes 0.56 mm (0.022 in.) If there were two such fitments on the part, as illustrated in Fig. 2-13c, and a center-to-center distance of 76.20 ± 0.13 mm (3:000 ± 0.005 in.) between them were to be applied, an additional clearance of 0.25 mm (0.010 in.) would have to be added for a total clearance of 0.81 mm (0.032 in.). That would be an unacceptably loose fit for many applications. Yet, a tolerance of ±0.13 mm (±0.005 in.) on a 25.40 mm (1 in.) dimension is quite tight for many high shrinking plastics. For the 76.20 mm (3 in.) dimension, that would be an extremely tight tolerance indeed.

The immediate reaction of the neophyte designer is to tighten the tolerances, thereby dramatically increasing the cost of both the part and the mold. Such a design change might even result in a respecification to a much more costly material. That, in effect, is fighting the material and the process instead of working with it. Experienced plastics engineers learn to use the inherent advantages of plastics to devise ways of fitting parts together using looser and less costly tolerances.

2.7   Design Practices for Looser Tolerances in Plastics

To design for the loosest tolerances compatible with desired performance, the first principle entails taking advantage of the properties plastics can provide. The most important one in this respect is the ability to alter the rigidity of the material such that one of the mating parts can be more rigid than the other. This property allows the more rigid of the two parts to cause the other part to conform to its contour, as illustrated in Fig. 2-14. The greater the difference in rigidity, the easier it is to accomplish this. Normally, the internal fitment is the more rigid. However, it is possible to specify the higher rigidity for the outer filaments – although usually there is less latitude.