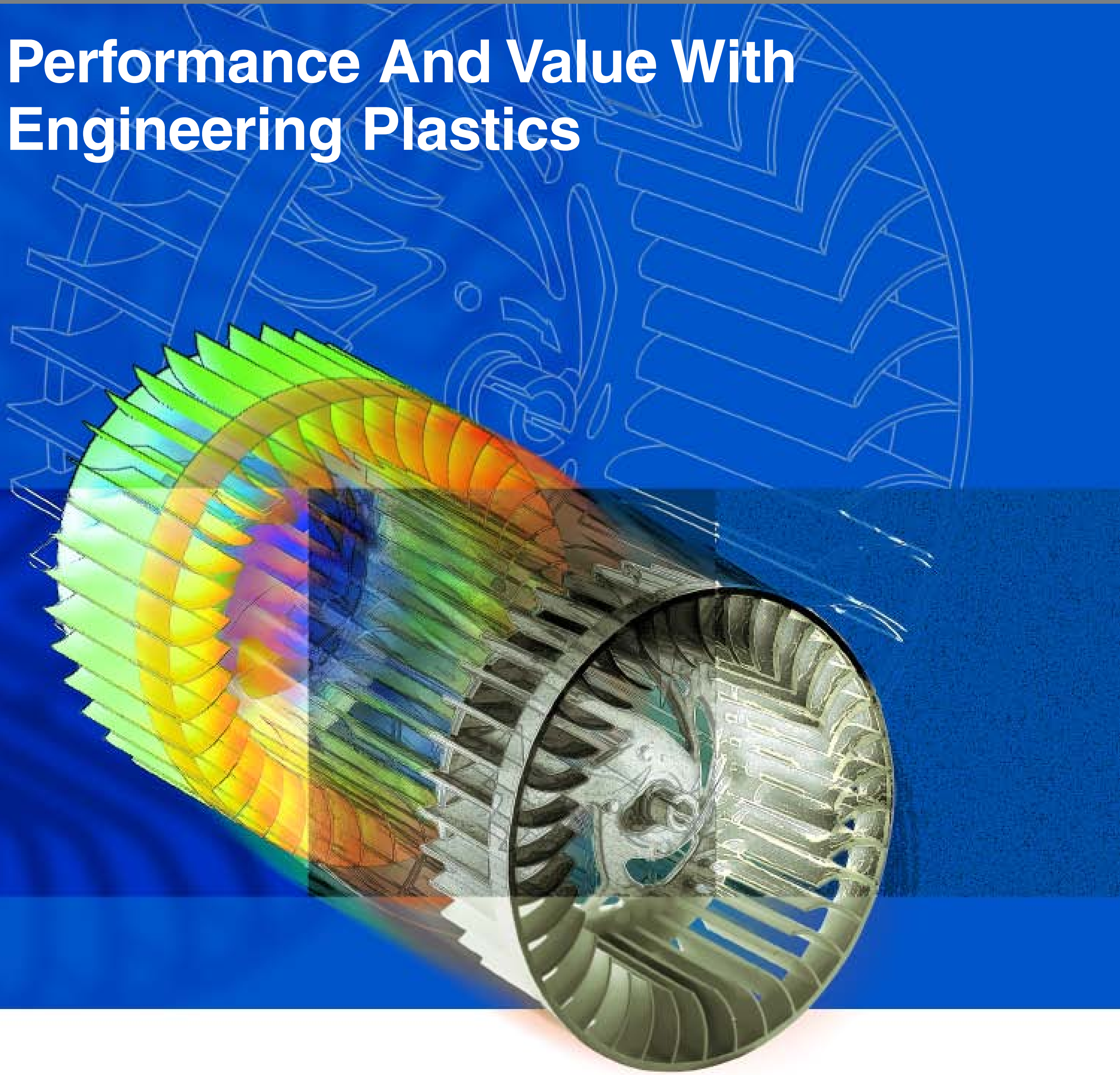
Design Guide

Performance And Value With Engineering Plastics

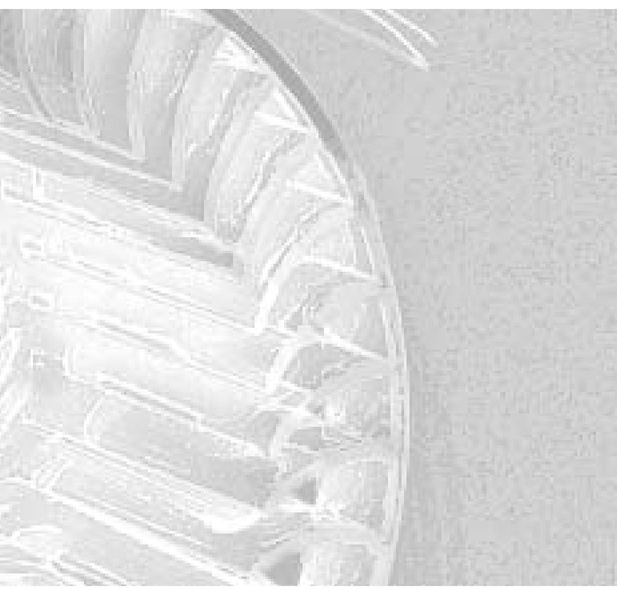


a DSM Guide

Unlimited. DSM

Unlimited. DSM

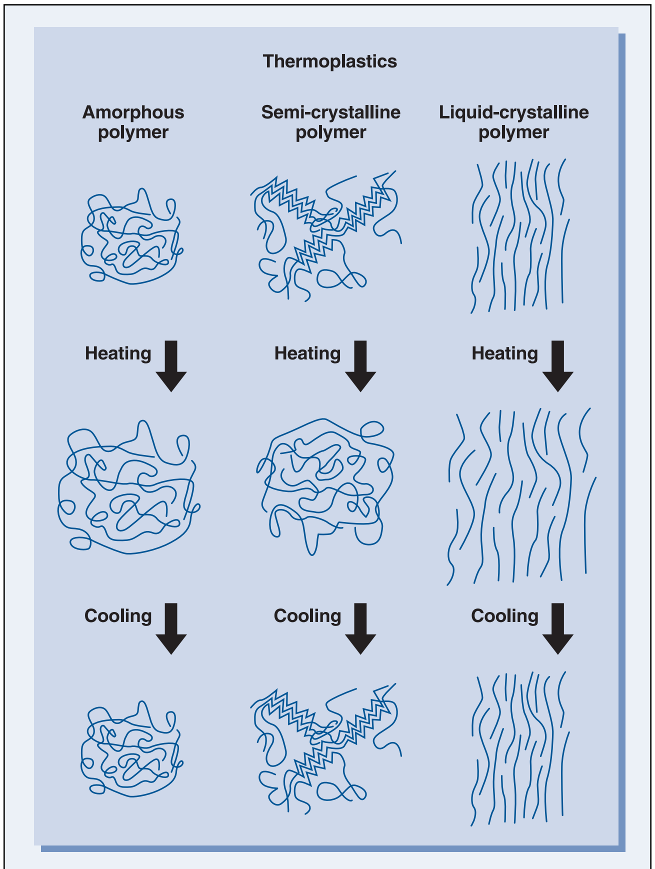
Table of Contents



Unlimited. DSM

Overview of Design Principles

Figure 1 Molecular structure of thermoplastics.



The process of developing thermoplastic parts requires a full understanding of typical material properties under various conditions. Thermoplastics can be categorized by their molecular structure as either amorphous, semi-crystalline plastics, or liquid crystal polymers (LCPs). The microstructures of these plastics and the effects of heating and cooling on the microstructures are shown in Figure 1.

Amorphous thermoplastics. Amorphous polymers have a structure that shows no regularity. In an unstressed molten state polymer molecules are randomly oriented and entangled with other molecules. Amorphous materials retain this type of entangled and disordered molecular configuration regardless of their states. Only after heat treatment some small degree of orientation can be observed (physical aging).

When the temperature of the melt decreases, amorphous polymers start becoming rubbery. When the temperature is further reduced to below the glass transition temperature, the amorphous polymers turn into glassy materials. Amorphous polymers possess a wide softening range (with no distinct melting temperature), moderate heat resistance, good impact resistance, and low shrinkage.

Figure 2 Amorphous versus semi-crystalline thermoplastics.

Polymer Type	Amorphous thermoplastics	Semi-crystalline thermoplastics
Tradename	Xantar polycarbonate Xantar C PC/ABS blend Stapron E PC/polyester blend	Akulon® PA 6 & 66 Stanyl® PA 46 Arnite® PBT & PET Arnitel® TPE-E's
Microstructure	Random molecular orientation in both molten and solid phases	Random molecular orientation in molten phase, but densely packed crysallites occrus in solid phase
Reaction to heat	Softens over a wide temperature range	Distinct melting temperature
General Properties		
Transparency	Transparent	Translucent or opaque
Specific gravity	Lower	Higher
Tensile strength	Lower	Higher
Tensile modules	Lower	Higher
Ductility	Higher	Lower
Resistance to creep	Lower	Higher
Fatigue performance	Lower	Higher
Max usage temp	Lower	Higher
Flow	Lower	Higher
Shrinkage & Warpage	Lower	Higher
Chemical resistance	Lower	Higher
Dimensional stability	Higher	Lower
Surface appearance	Higher	Lower

Semi-crystalline thermoplastics. Semi-crystalline plastics, in their solid state, show local regular crystalline structures dispersed in an amorphous phase. These crystalline structures are formed when semi-crystalline plastics cool down from melt to solid state. The polymer chains are partly able to create a compacted structure with a relatively high density. The degree of crystallization depends on the length and the mobility of the polymer segments, the use of nucleants, the melt, and the mold temperatures.

Liquid crystal polymers. Liquid crystal polymers (LCPs) exhibit ordered molecular arrangements in both the melt and solid states. Their stiff, rod-like molecules that form the parallel arrays or domains characterize these materials.

The difference in molecular structure may cause remarkable differences in properties. Various properties are time or temperature dependent. The shear modulus, for instance, decreases at elevated temperatures. The shear modulus curve illustrates the temperature limits of a thermoplastic. The shape of the curve is different for amorphous and semi-crystalline thermoplastics (see Figure 3)

Figure 3 Glass transition temperature (Tg) and melt temperature (Tm).

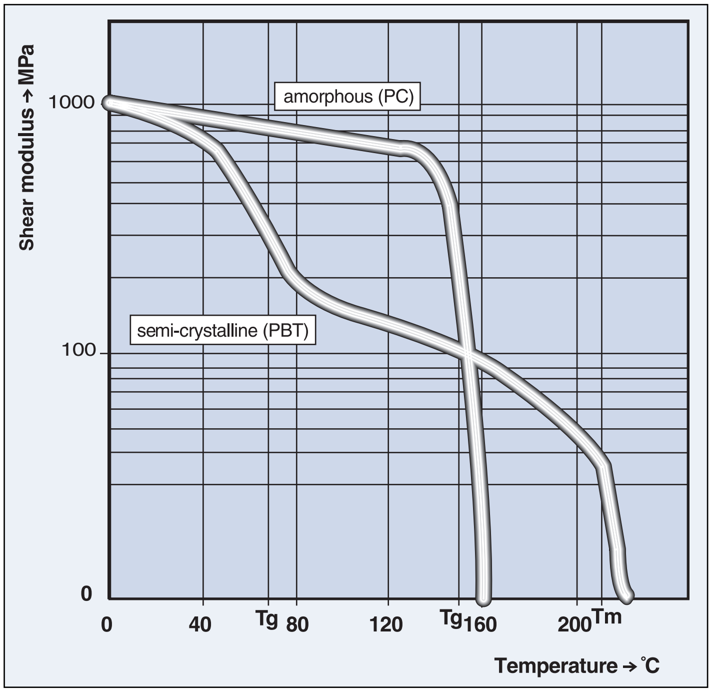
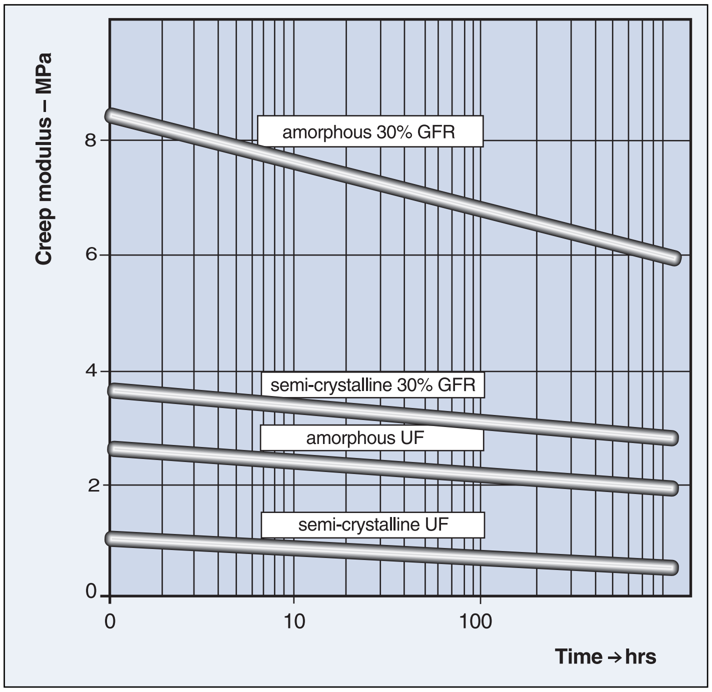


Figure 4 The following graph demonstrates time dependent creep moduli. In general semi-crystalline materials have lower creep rates than amorphous materials. Glass reinforcement generally improves the creep resistance of a thermoplastic material.



Unlimited. DSM

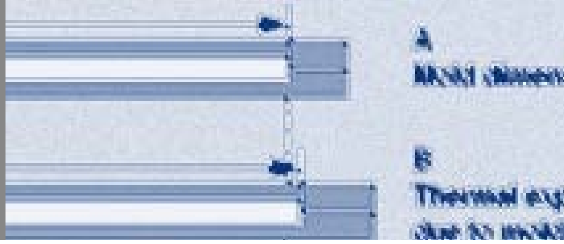


Figure 5 Shrinkage indication of DSM polymers in %.

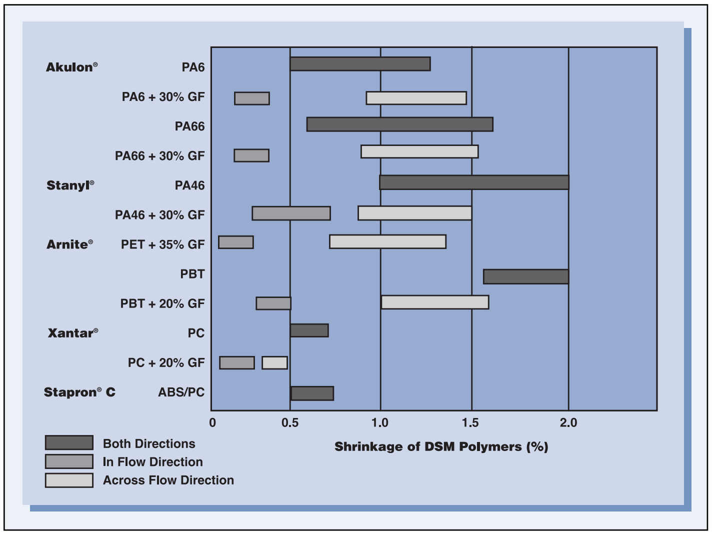
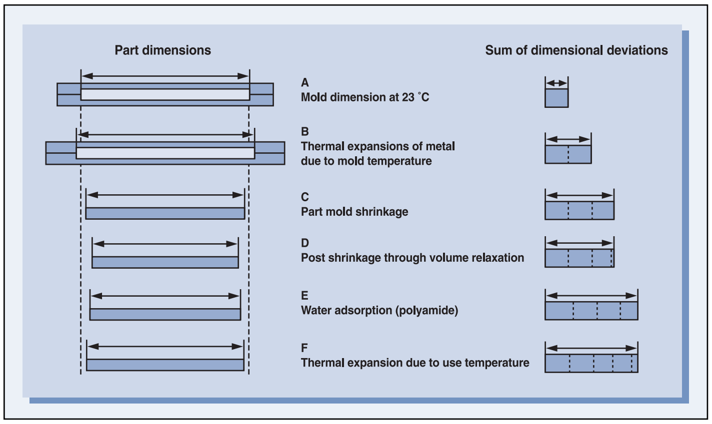


Figure 6 Dimensional stability through time %.



Dimensional Stability

The following information on mold shrinkage, thermal expansion, and water absorption relates to the precision of a component both during molding and as secondary effects after molding.

Mold shrinkage. Shrinkage or mold shrinkage is the difference between the mold cavity dimensions and the corresponding component dimensions. It is not possible to predict exact shrinkage values for a specific polymer grade. Therefore, the maximum and minimum values for the various DSM thermoplastics are provided in Figure 5.

During injection molding the polymer melt is injected into the mold. Once the mold is completely filled the dimensions of the molding are the same as the dimensions of the mold cavity at its service temperature (see "B" in Figure 6). While cooling down, the polymer starts to shrink (see "C" in Figure 6). During the holding stage of the injection molding cycle, shrinkage is compensated by post-filling/packing. Both the design of the part as well as the runner/gate should allow for sufficient filling and packing.

The process of shrinkage continues even after the part has been ejected. Shrinkage should be measured long enough after injection molding to take into account post-shrinkage (see "D" in Figure 6).

Secondary effects. If components are heated after molding, for example during paint curing operation, this can cause temporary or even permanent dimensional changes. The operating environment will also have consequences for the dimensional stability of the component.

Thermal expansion. An important condition for the dimensions of a part is the use temperature. Thermoplastics show a relatively high thermal expansion (10^{-4}/ °C) compared to metals (10^{-5}/ °C). Thermal expansion cannot be ignored for large parts that are used at elevated temperatures (see "F" in Figure 6).

Moisture absorption. Akulon and Stanyl parts, like all polyamide moldings, show dimensional changes increase after molding due to moisture absorption (see Figure 7). Moisture absorption is a time dependent, reversible process that continues until equilibrium is reached. This equilibrium depends on temperature, relative humidity of the environment and the wall thickness of the molding.

A change in moisture content will result in different product dimensions. The designer should anticipate varying humidity conditions during use of the product (see E in Figure 6). The moisture absorption of reinforced grades differs from those of the unfilled grades.

The moisture content not only affects the dimensions but also various important properties. Yield stress, modulus of elasticity and hardness decrease with increasing moisture absorption, while toughness shows a considerable increase.

Although polyamide moldings are already comparatively tough in the dry state, the high toughness, which is characteristic of Akulon and Stanyl, is not reached until the material has absorbed 0.5-1 moisture. Unreinforced Stanyl already shows a dry as molded impact resistance twice as high as other polyamides, so conditioning is less critical.

Example of dimensional stability. Examples of the dimensional stability of unfilled and reinforced Akulon and Stanyl are shown in Figure 8. For polyamide grades in general, the swelling of the thickness is substantial, especially when compared to the swelling in the two other directions. This should be taken into account when designing parts with thick walls.

Dimensional deviations/tolerances. All factors discussed influence the final dimensions of the part. The maximum dimensional deviation of the part is the sum of the individual contributing factors (see Figure 6).

Figure 7 Effect of time and humidity on moisture absorption.

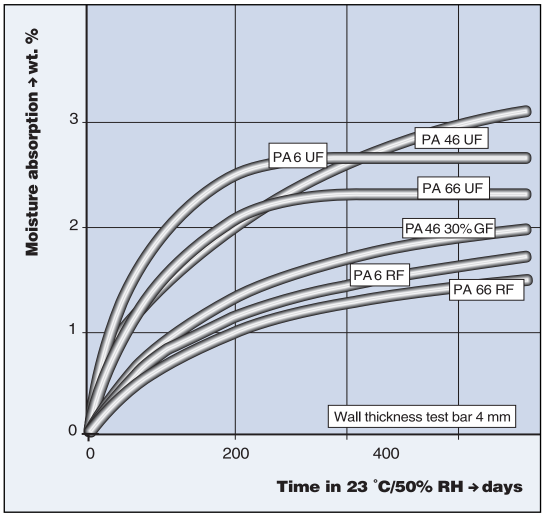
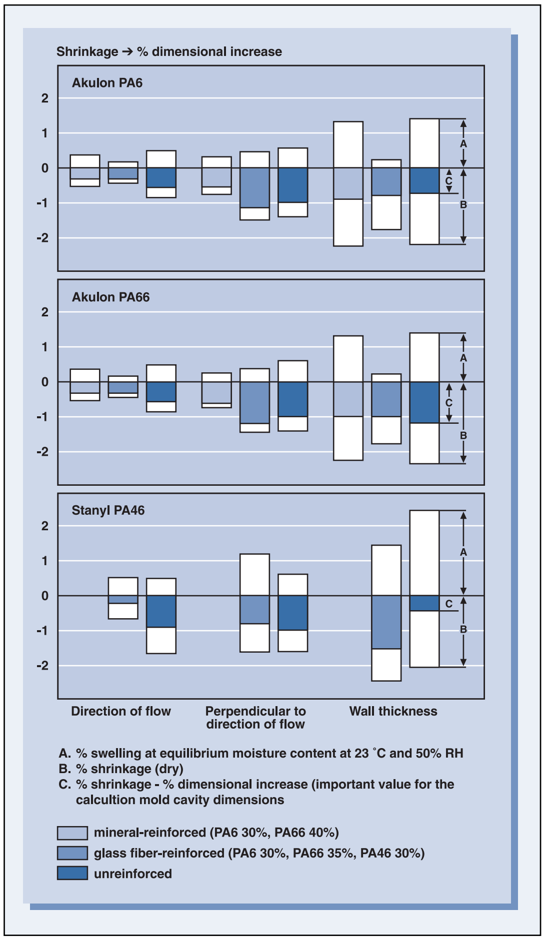


Figure 8 Dimensional stability of Akulon PA6/PA66 and Stanyl PA46.



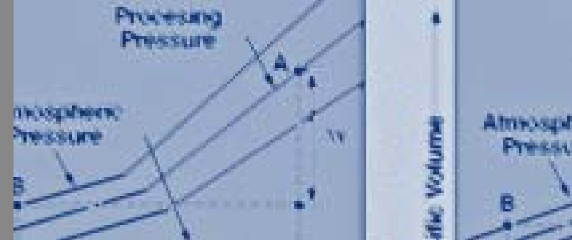


Figure 9 pvT curves for amorphous and crystalline polymers.

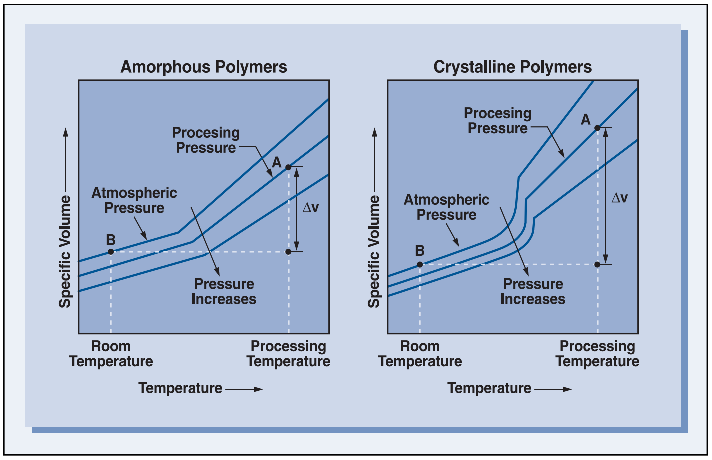
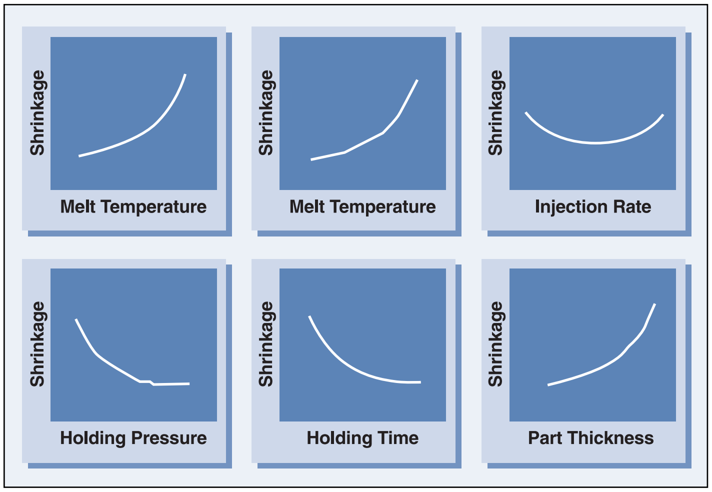


Figure 10 Processing and design parameters that affect part shrinkage.



Shrinkage

Shrinkage is inherent in the injection molding process. Shrinkage occurs because the density of polymer varies from the processing temperature to the ambient temperature. The shrinkage of ed plastic parts can be as much as 20 percent by volume, when measured at the processing temperature and the ambient temperature.

Semi-crystalline materials are particularly prone to thermal shrinkage; amorphous materials tend to shrink less. When crystalline materials are cooled below their transition temperature, the molecules arrange themselves in a more orderly way, forming crystallites. On the other hand, the microstructure of amorphous materials does not change with the phase change. This difference leads to semi-crystalline materials having a greater difference in specific volume between the processing state (point A) and the state at room temperature and atmospheric pressure (point B) than amorphous materials (seen Figure 9).

During injection molding, the variation in shrinkage both globally and through the cross section of a part creates internal stresses, (residual stresses). If the residual stresses are high enough to overcome the structural integrity of the part, the part will warp upon ejection from the mold or crack with external service load.

Uncompensated volumetric contraction leads to either sink marks or voids in the interior of the part. Shrinkage that leads to sink marks or voids can be reduced or eliminated by packing the cavity after filling. Controlling part shrinkage is particularly important in applications requiring tight tolerances.

Excessive shrinkage can be caused by a number of factors:

- Low effective holding pressure
- Short pack-hold time or cooling time
- Fast freezing off of gate
- High melt temperature
- High temperature

The relationship of shrinkage to several processing parameters and part thickness is shown schematically in Figure 10.

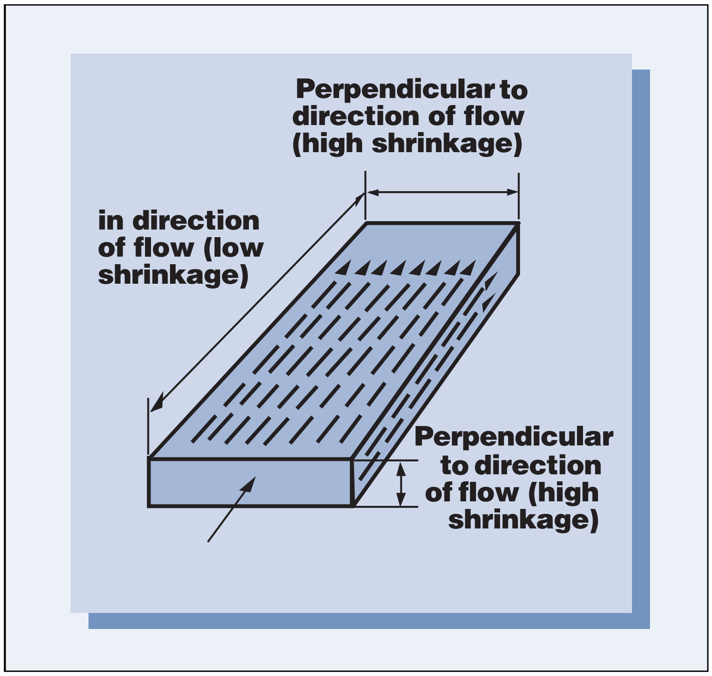
Isotropic versus anisotropic shrinkage. For both unfilled amorphous and mineral-reinforced thermoplastics shrinkage is largely isotropic; shrinkage in flow direction is about equal to the shrinkage across flow. The glass fiber reinforced grades, on the other hand, show anisotropic properties. Due to fiber orientation in the direction of the melt flow, shrinkage values in flow direction often are substantially smaller than across flow direction (see Figure 11).

The assumption a material has isotropic properties is often a good starting point but if anisotopy is totally ignored significant errors can occur in the design of thermoplastic parts.

The designer must be aware that as the degree of anistopy increases so does the number of moduli required to describe the material, up to a maximum of 21. Therefore especially for glass-reinforced materials the use of simple analysis techniques has limited value and extensive FEA (finite element analysis) methods are often required to analyze anistropic materials in critical applications.

Not only fiber orientation but also molecular orientation can lead to anistropic shrinkage. An unfilled molded part containing high levels of molecular orientation, due to high shear stresses, can show anistropic shrinkage because aligned chains shrink more in the direction of orientation.

Figure 11 Relation between the shrinkage of glass fiber reinforced plastics and the orientation of the glass fibers (in thickness direction).



Design Guidelines

Figure 12 Spiral flow length of Akulon Ultraflow at 260°C and 1400 bar.

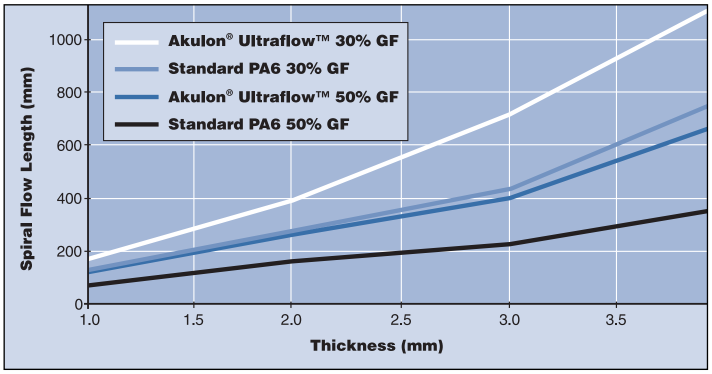
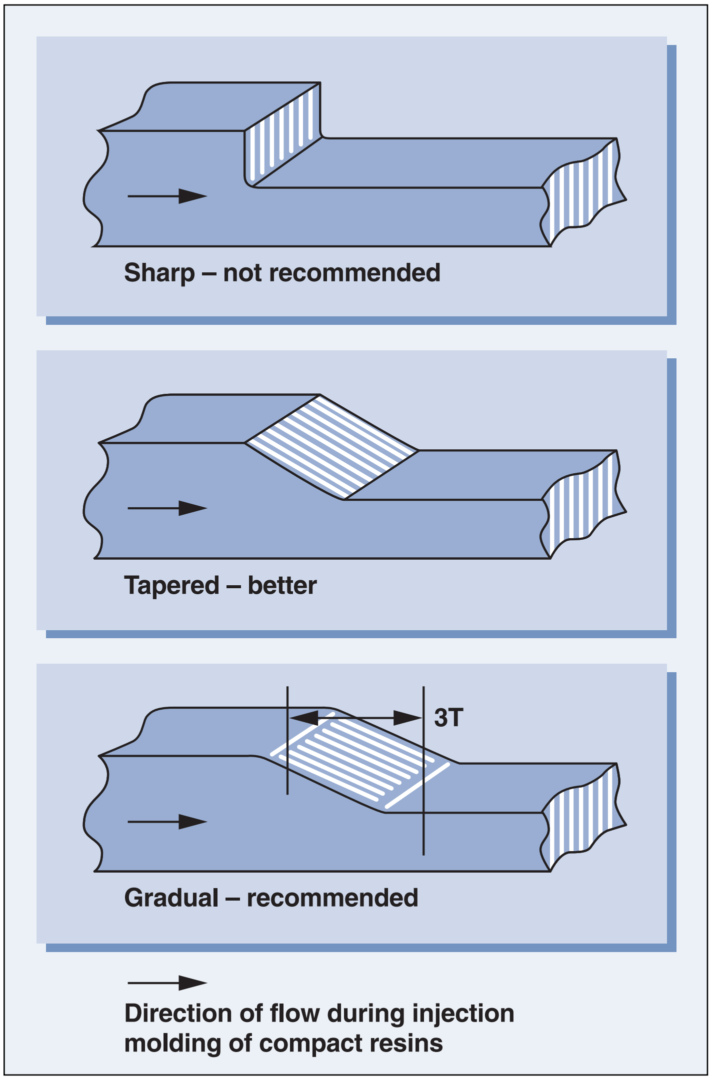


Figure 13 Gradual transition of wall thicknesses.



Wall Thickness

Just as metals have normal working thickness ranges based upon their processing method, so do plastics. Typically, for injection molded parts, the wall thickness will be in the range 0.5 mm to 4 mm (0.20 - 0.16 in). Dependent on the part design and size, parts with either thinner or thicker sections can be molded.

While observing functional requirements, keep wall thicknesses as thin and uniform as possible. In this way even filling of the mold and anticipated shrinkage throughout the molding can be obtained in the best way. Internal stresses can be reduced.

Wall thickness should be minimized to shorten the molding cycle, obtain low part weight, and optimize material usage. The minimum wall thickness that can be used in injection molding depends on the structural requirements, the size and geometry of the molding, and the flow behavior of the material. As a starting point the designer can often refer to spiral flow curves which give a relative measure of the maximum achievable flow length for a given wall thickness and injection pressure. See Figure 12.

If parts are subjected to any significant loading the part should be analyzed for stress and deflection. If the calculated stress or deflection value is not acceptable a number of options could be considered including the following:

- Increase wall (if not already too thick)
- Use an alternative material with higher strength and/or modulus
- Incorporate ribs or contours in the design to increase the sectional modulus

Other aspects that may need to be considered include:

Insulation characteristics
Generally speaking insulating ability (whether for electrical or heat energy) is related to the thickness of the polymer.

Impact characteristics
Impact resistance is directly related to the ability of a part to absorb mechanical energy without fracturing. This in turn is related to the part design and polymer properties. Increasing the wall section will generally help with impact resistance but too thick (stiff) a section may make a design unable to deflect and distribute an impact load therefore increasing stresses to an unacceptable level.

Agency approval
When a part design must meet agency requirements for flammability, heat resistance, electrical properties etc, it may be necessary to design with thicker sections than would be required just to meet the mechanical requirements.

Where varying wall thicknesses are unavoidable for reasons of design, there should be a gradual transition (3 to 1) as indicated in Figure 13.

Generally, the maximum wall thickness used should not exceed 4 mm (0.16 in). Thicker walls increase material consumption, lengthen cycle time considerably, and cause high internal stresses, sink marks and voids (see Figure 14).

Care should be applied to avoid a "race tracking" effect, which occurs because melt preferentially flows faster along thick sections. This could result in air traps and welds lines, which would appear as surface defects. Modifying or incorporating ribs in the design can often improve thick sections.

Figure 14 Sink marks due to large wall thicknesses.

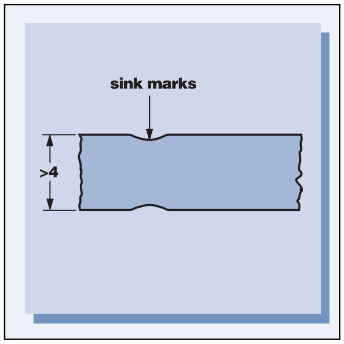


Figure 15 Voids due to large wall thicknesses.

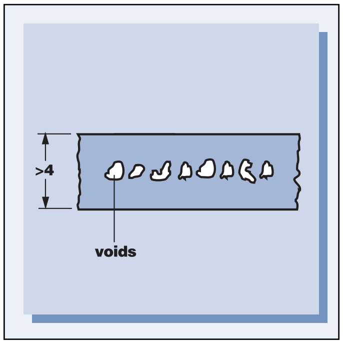


Figure 16 Influence of rib design on flow behavior of the melt.

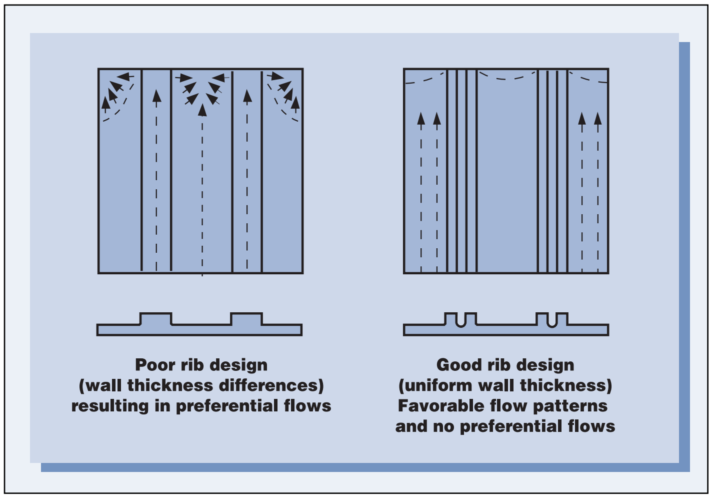


Figure 17 Example of design study of multi-connector.

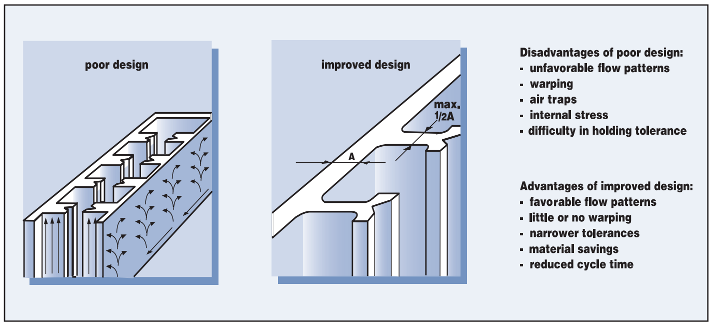


Figure 18 Differential shrinkage for filled and unfilled materials.

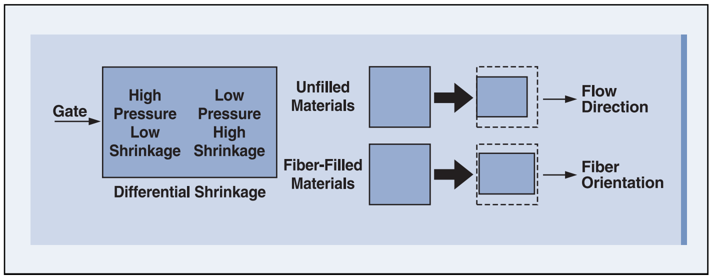


Figure 19 Unreinforced vs fiber reinforced materials.

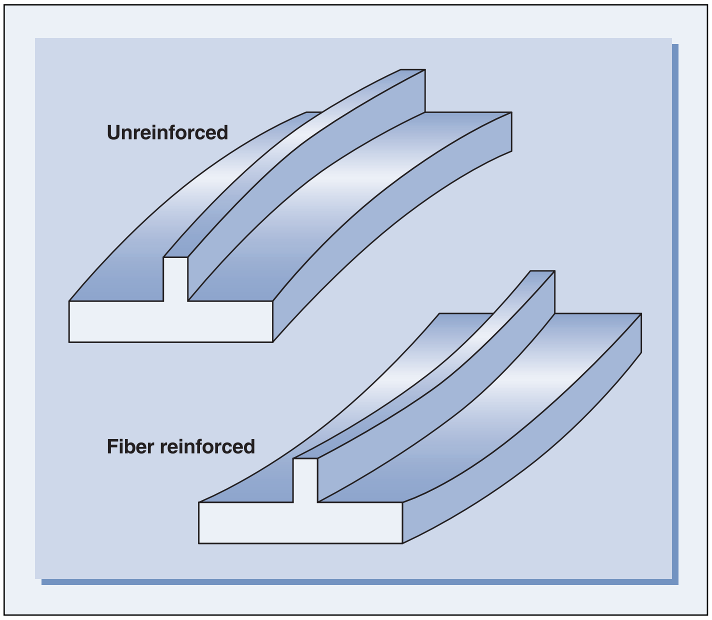
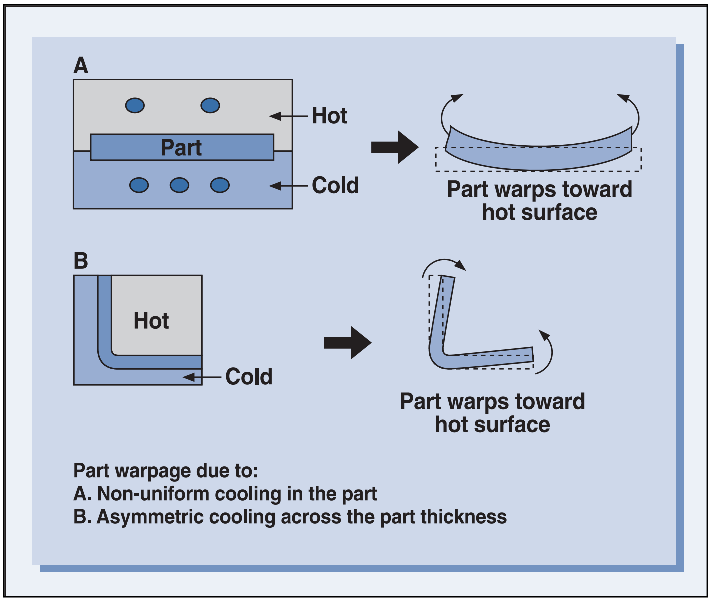


Figure 20 Part warpage due to: (a) non-uniform cooling in the part and (b) asymmetric cooling across the part thickness.



Warpage

If the shrinkage throughout the part is uniform, the molding will not deform or warp; it simply becomes smaller. Warpage can be considered as a distortion where the surfaces of the molded part do not follow the intended shape of the design.

Part warpage results from differential shrinkage in the material of the molded part caused by molded-in residual stresses. Variations in shrinkage can be caused by molecular and fiber orientation, temperature variations within the molded part, and by variable packing, such as over-packing at gates and under-packing at remote locations, or different pressure levels as material solidify across the part thickness. Due to the number of factors present it is a very complicated task to achieve uniform shrinkage. These causes are described more fully in the information that follows.

Influence of unfilled and filled materials. For fiber reinforced thermoplastics, reinforcing fibers inhibit shrinkage due to their smaller thermal contraction and higher modulus. Therefore, fiber reinforced materials shrink less along the direction in which fibers align (typically the flow direction) compared to the shrinkage in the transverse direction. Similarly, particle-filled thermoplastics shrink less than unfilled grades, but exhibit a more isotropic nature. For non-reinforced materials warpage is generally influenced by wall thickness and mold temperature. If wall thickness and mold temperatures are not optimal the molding will most likely warp.

For glass reinforced materials totally different characteristics are evident due to fiber orientation. If a non-reinforced and a fiber reinforced material are compared in the same design it is possible to see contrary warpage in the same part.

Influence of cooling. Non-uniform cooling in the part and asymmetric cooling across the part thickness from the cavity and core can also induce differential shrinkage. The material cools and shrinks inconsistently from the wall to the center, causing warpage after ejection.

Influence of wall thickness. Shrinkage increases as the wall thickness increases. Differential shrinkage due to non-uniform wall thickness is a major cause of part warpage in unreinforced thermoplastics. More specifically, different cooling rates and crystallization levels generally arise within parts with wall sections of varying thickness. Larger volumetric shrinkage due to the high crystallization level in the slow cooling areas leads to differential shrinkage and thus part warpage.

Influence of asymmetric geometry. Geometric asymmetry (e.g., a flat plate with a large number of ribs that are aligned in one direction or on one side of the part) will introduce non-uniform cooling and differential shrinkage that can lead to part warpage. The poor cooling of the wall on the ribbed side causes a slower cooling of the material on that one side, which can lead to part warpage.

Figure 21 Diagram of high shrinkage/low cooling vs warped part.

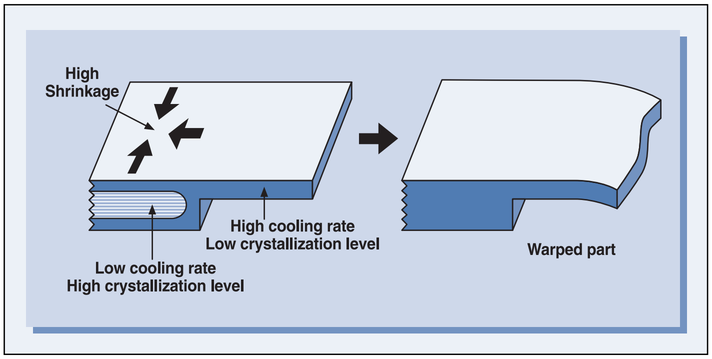
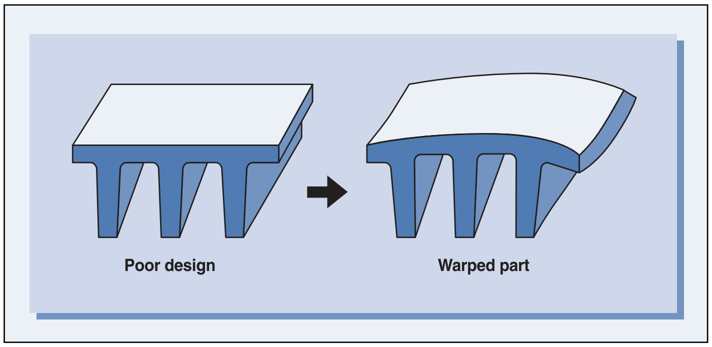


Figure 22 Poor design vs warped part.



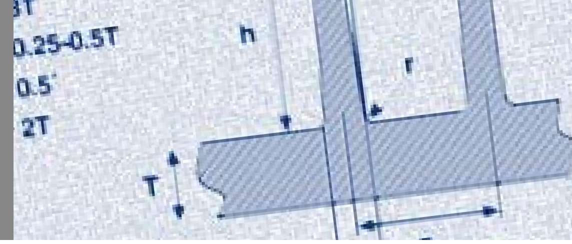


Figure 23 Recommendations for rib dimensions.

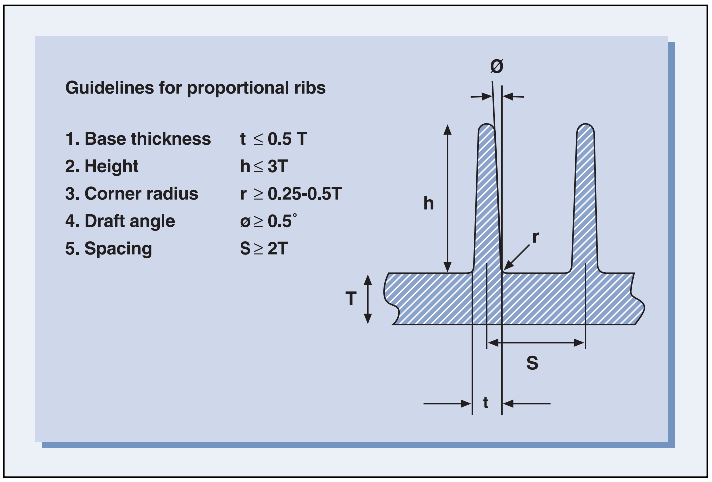


Figure 24 Ribs.

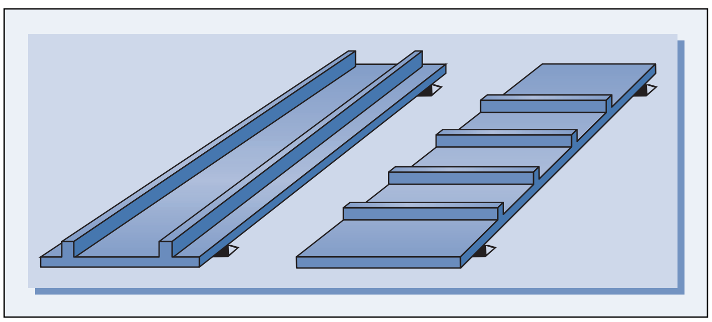
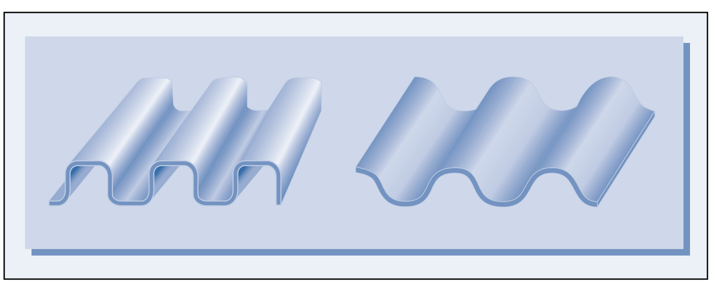


Figure 25 Corrugations.



Ribs and Profiles Structures

If the load carrying ability or the stiffness of a plastic structure needs to be improved it is necessary to either increase the sectional properties of the structure or change the material. Changing the material or grade of material, e.g. higher glass fiber content, may be adequate sometimes but is often not practical (different shrinkage value) or economical.

Increasing the sectional properties, namely the moment of inertia, is often the preferred option. As discussed in other sections, just increasing the wall section although the most practical option will be self-defeating.

- Increase in part weight and costs are proportional to the increase in thickness.
- Increase in cooling time is proportional to the square of the increase in thickness.

If the load on a structural part requires sections exceeding 4 mm (0.16 in) thickness, reinforcement by means of ribs or box sections is advisable in order to obtain the required strength at an acceptable wall thickness.

Solid plate vs. ribbed plate in terms of weight and stiffness. Although ribs offer structural advantages they can give rise to warpage and appearance problems, for this reason certain guidelines should be followed:

- The thickness of a rib should not exceed half the thickness of the nominal wall as indicated in Figure 23.
- In areas where structure is more important than appearance, or with very low shrinkage materials, ribs with a thickness larger than half the wall thickness can be used. These will cause sink marks on the surface of the wall opposite the ribs. In addition, thick ribs may act as flow leaders causing preferential flows during injection. This results in weld lines and air entrapment.

- Maximum rib height should not exceed 3 times the nominal wall thickness as deep ribs become difficult to fill and may stick in the mold during ejection.
- Typical draft is 1 to 1.5 deg per side with a minimum of 0.5 deg per side. Generally draft and thickness requirements will limit the rib height.
- At the intersection of the rib base and the nominal wall a radius of 25 to 50% of the nominal wall section should be included. Minimum value 0.4 mm. This radius will eliminate a potential stress concentration and improve flow and cooling characteristics around the rib. Larger radii will give only marginal improvement and increase the risk of sink marks on the opposite side of the wall.

Parallel ribs should be spaced at a minimum distance of twice the nominal wall thickness; this helps prevent cooling problems and the use thin blades in the mold construction.

Ribs are preferably designed parallel to the melt flow as flow across ribs can result in a branched flow leading to trapped gas or hesitation. Hesitation can increase internal stresses and short shots.

Parallel ribs should be spaced at a minimum distance of twice the nominal wall thickness; this helps prevent cooling problems and the use thin blades in the mold construction. Ribs are preferably designed parallel to the melt flow as flow across ribs can result in a branched flow leading to trapped gas or hesitation. Hesitation can increase internal stresses and short shots.

Ribs should be orientated along the axis of bending in order to provide maximum stiffness. Consider the example in Figure 24 where a long thin plate is simply supported at the ends. If ribs are added in the length direction the plate is significantly stiffened. However, if ribs are added across the width of the plate little improvement is found.

Figure 26 Flat and open areas.

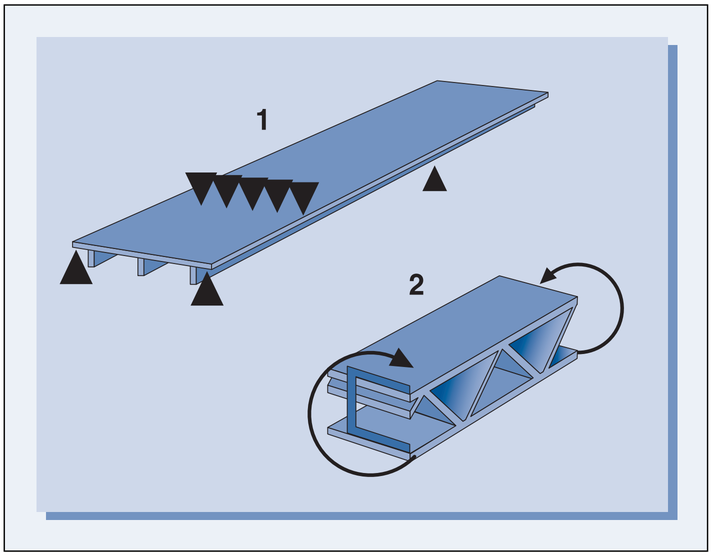
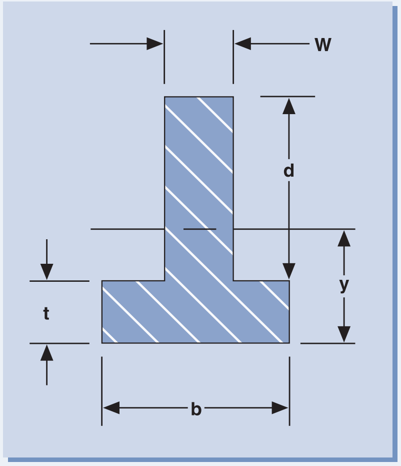
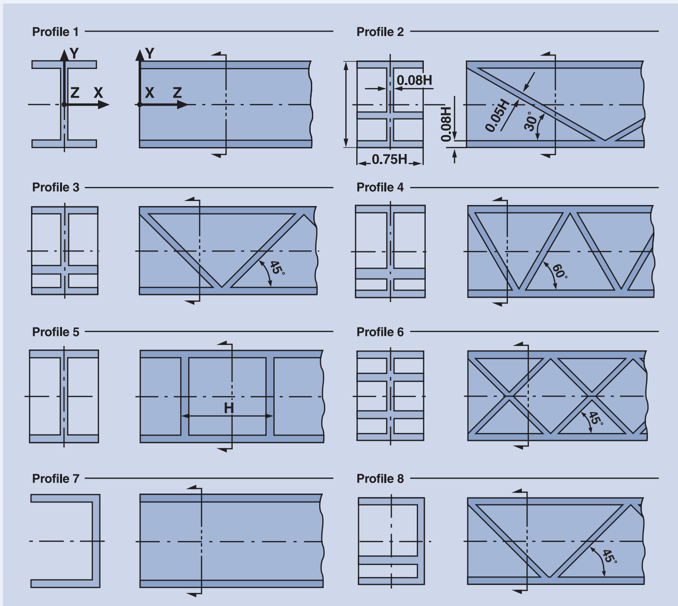


Figure 27 Dimension image with chart of case 1-6.



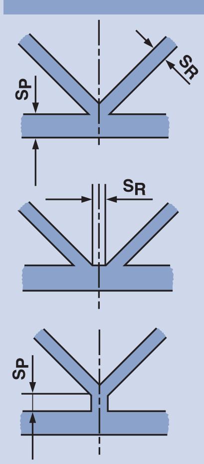
Dimension	Case 1	Case 2	Case 3	Case 4	Case 5	Case 6
b (mm)	40	40	40	40	40	40
t (mm)	4	8	4	4	4	4
d (mm)	0	0	3	6	9	12
w (mm)	0	0	2	2	2	2
Area mm^2	160	320	166	172	178	184
y (mm)	2	4	2.12	2.35	2.65	3.04
Momentum of inertia	213.3	1706.7	288.7	528.4	1018.4	1837.0
% increase stiffness	NA	700	35.3	147.7	377.4	761.1
% increase weight	NA	100	3.75	7.5	11.25	15

Figure 28 Comparison of profiles in terms of torsional rigidity and bending.



Profile	1	2	3	4	5	6	7	8
Torsional rigidity	1	10	17	19	1	34	1	37
Horizontal bending	1	1.1	1	1	1.1	1.1	1.9	2
Vertical bending	1	1.1	1.1	1.1	1.1	1.1	1	1

Figure 29 Torsional rigidity and resistance to torsional stress as a function of the way in which the ribs are connected to the profile.

Variants	Rigidity in %	Resistance to stress in %
	100	100
	65	80
	60	82

Ratio Sr/Sp = 0.69

Ribbing is typically applied for:

- Increasing bending stiffness or strength of large flat areas
- Increasing torsional stiffness of open sections

Adding corrugations to the design can stiffen flat surfaces in the direction of the corrugations (see Figure 25). They are very efficient and do not add large amounts of extra material or lengthen the cooling time. The extra stiffness is a result of increasing the average distance of the material from the neutral axis of the part, i.e. increasing the second moment of inertia.

Ribs and box sections increase stiffness, thus improving the load bearing capability of the molding. These reinforcing methods permit a decrease in wall thickness but impart the same strength to the section as a greater wall thickness.

The results demonstrate that the use of diagonal ribs have the greatest effect on the torsional rigidity of the section. The change from an I section to a C section helps in terms of horizontal bending terms but not in torsional terms. As double cross ribs (Figure 28 - option 6) can give tooling (cooling) problems option 8 is the recommended solution for the best torsional performance.

Depending on the requirements of the part the acceptability of possible sink marks at the intersection of the ribs and profile wall need special consideration. For maximum performance and function the neutral lines of the ribs and profile wall should meet at the same point. Deviation from this rule will result in a weaker geometry. If, due to aesthetic requirements, the diagonal ribs are moved slightly apart then the rigidity is reduced 35%. If a short vertical rib is added to the design then the torsional rigidity is reduced an additional 5% (see Figure 29).

Gussets or Support Ribs

Gussets can be considered as a subset of ribs and the guidelines that apply to ribs are also valid for gussets. This type of support is used to reinforce corners, side walls, and bosses.

The height of the gusset can be up to 95% of the height of the boss or rib it is attached to. Depending on the height of the rib being supported gussets may be more than 4 times the nominal wall thickness. Gusset base length is typically twice the nominal wall thickness. These values optimize the effectiveness of the gusset and the ease of molding and ejecting the part.

Bosses

Bosses often serve as mounting or fastening points and therefore, for good design, a compromise may have to be reached to achieve good appearance and adequate strength. Thick sections need to be avoided to minimize aesthetic problems such as sink marks. If the boss is to be used to accommodate self tapping screws or inserts the wall section must be controlled to avoid excessive build up of hoop stresses in the boss.

General recommendations include the following:

- Nominal boss wall thickness less than 75% nominal wall thickness, note above 50% there is an increased risk of sink marks. Greater wall sections for increased strength will increase molded-in stresses and result in sink marks.
- A minimum radius of 25% the nominal wall thickness or 0.4 mm at the base of the boss is recommended to reduce stresses.
- A minimum draft of 0.5 degrees is required on the outside dimension of the boss to ensure release from the mold on ejection.

Figure 30 Guidelines for gussets.

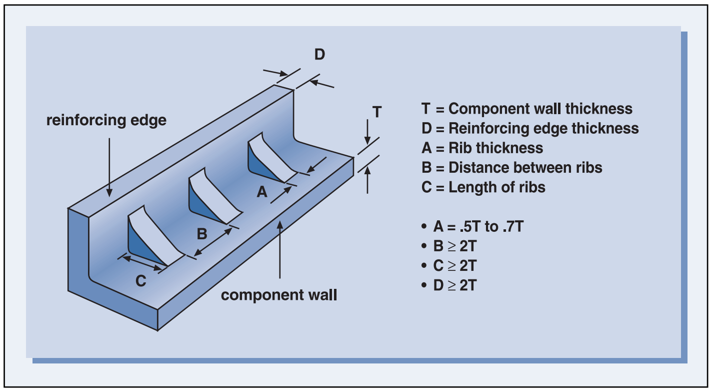


Figure 31 Height of the gusset.

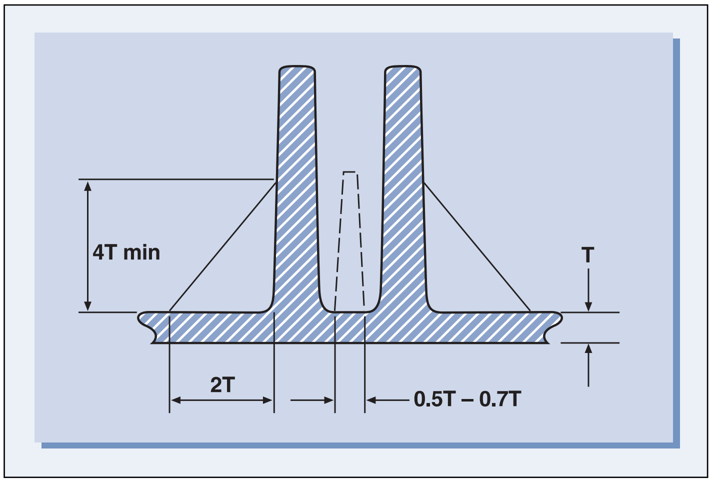


Figure 32 Proper boss design.

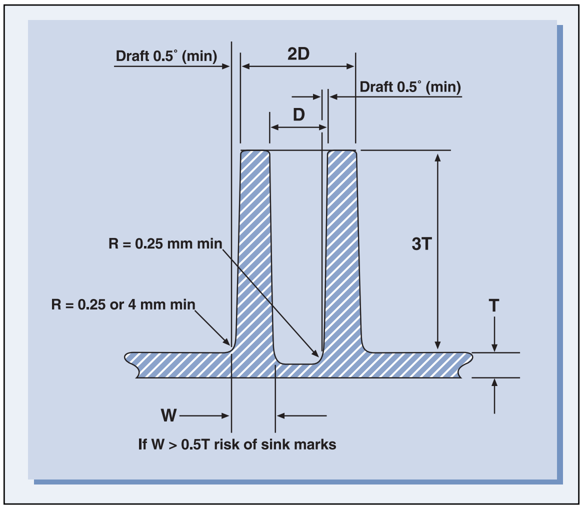


Figure 33 Correct positioning of bosses.

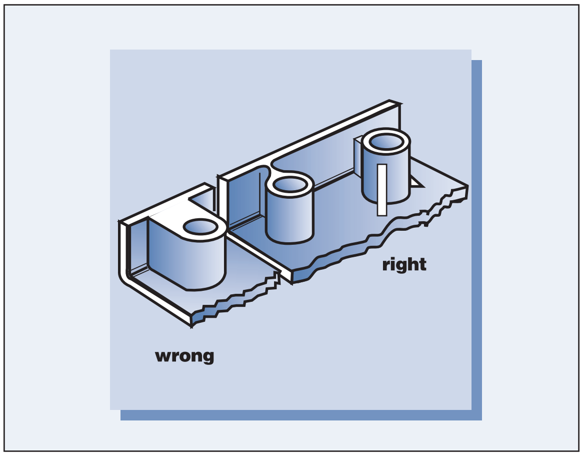
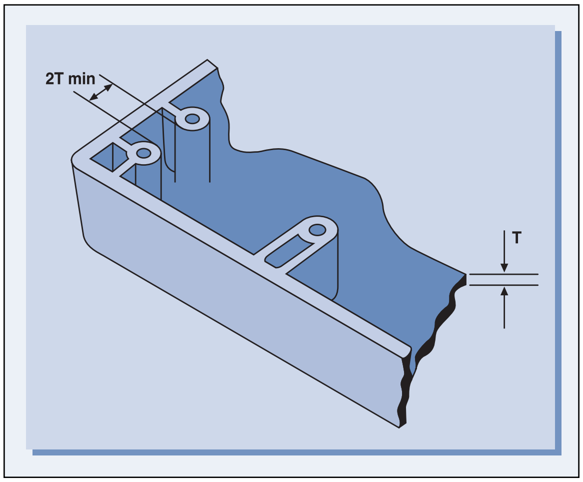


Figure 34 Boss spacing.



- Increasing the length of the core pin so that it penetrates the nominal wall section can reduce the risk of sink marks. The core pin should be radiused (min 0.25 mm) to reduce material turbulence during filling and to help keep stresses to a minimum. This option does increase the risk of other surface defects on the opposite surface.
- A minimum draft of 0.25 degrees is required on the internal dimension for ejection and or proper engagement with a fastener.

Further strength can be achieved with gusset ribs or by attaching the boss to a sidewall. Bosses adjacent to external walls should be positioned a minimum of 3 mm (.12 in) from the outside of the boss to avoid creating a material mass that could result in sink marks and extended cycle times (see Figure 33). A minimum distance of twice the nominal wall thickness should be used for determining the spacing between bosses (see Figure 34). If placed too close together thin areas that are hard to cool will be created. These will in turn affect quality and productivity.

Holes

Holes are easily produced in molded parts by core pins. Through holes are easier to produce than blind holes because the core pin can be supported at both ends.

Blind holes. Core pins supported by just one side of the mold tool create blind holes. The length of the pins, and therefore the depth of the holes, are limited by the ability of the core pin to withstand any deflection imposed on it by the melt during the injection phase. See information on bosses & cores. As a general rule the depth of a blind hole should not exceed 3 times the diameter. For diameters less than 5 mm this ratio should be reduced to 2.

Through holes. With through holes the cores can be longer as the opposite side of the mold cavity can support them. An alternative is to use a split core fixed in both halves of the mold that interlock when the mold is closed. For through holes the length of a given size core can be twice that of a blind hole. In cases where even longer cores are required, careful tool design is necessary to ensure balanced pressure distribution on the core during filling to limit deflection.

Figure 35 Blind cores.

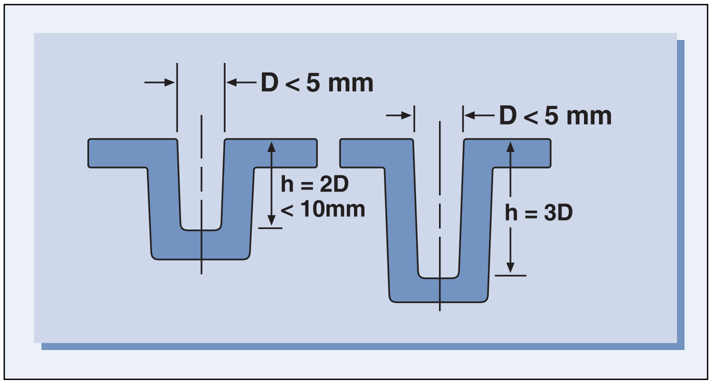


Figure 36 Through cores.

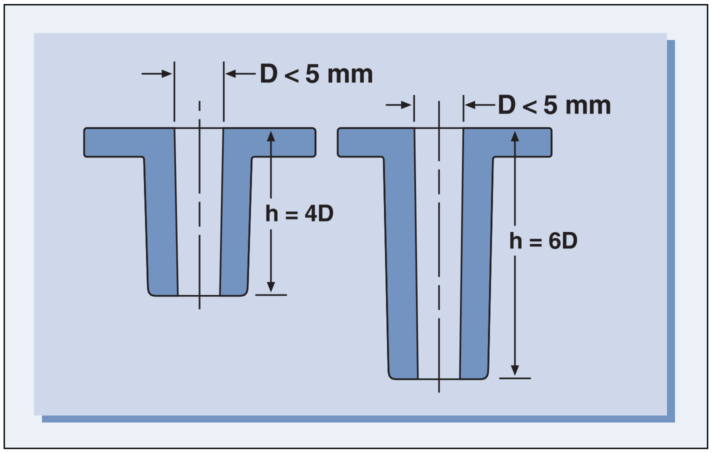


Figure 37 Minimum hole spacing dimensions.

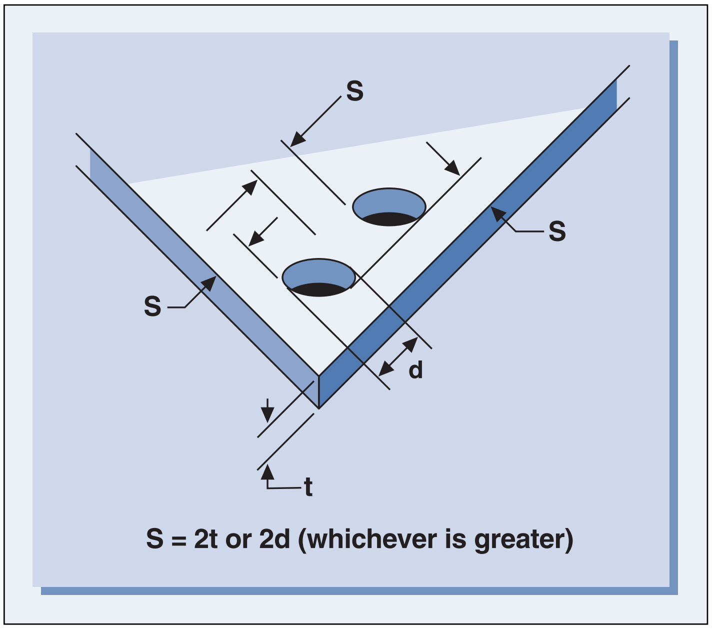
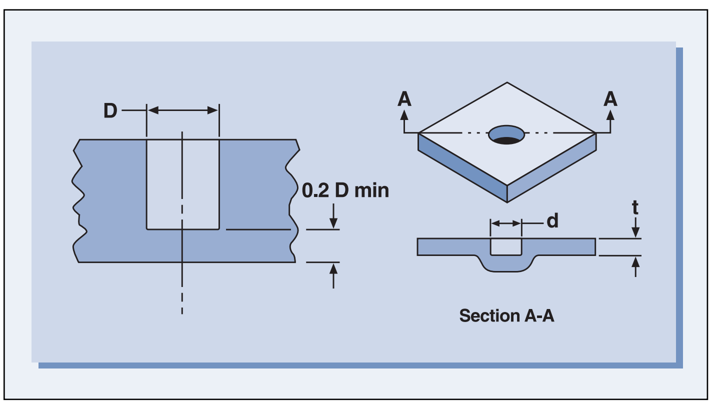


Figure 38 Blind hole design recommendations.



Holes with an axis that runs perpendicular to the mold opening direction require the use of retractable pins or split tools. In some designs placing steps or extreme taper in the wall can avoid this. See section on draft. Core pins should be draw polished and include draft to help with ejection. The mold design should direct the melt flow along the length of slots or depressions to locate weld lines in thicker or less critical sections. If weld lines are not permissible due to strength or appearance requirements, holes may be partially cored to facilitate drilling as a post molding operation. The distance between two holes or one hole and the parts edge should be at least 2 times the part thickness or 2 times the hole diameter whichever is the largest.

For blind holes the thickness of the bottom should be greater than 20% of the hole diameter in order to eliminate surface defects on the opposite surface. A better design is to ensure the wall thickness remains uniform and there are no sharp corners where stress concentrations can occur.

Radii & Corner

In the design of injection molded parts sharp corners should always be avoided; generous radii should be included in the design to reduce stress concentrations. Fillet radii should be between 25 and 60% of the nominal wall thickness. If the part has a load bearing function then the upper end is recommended. A minimum radius of 0.5 mm is suggested and all sharp corners should be broken with at least a 0.125 mm radius.

Sharp corners, particularly internal corners introduce:
- High molded in stresses
- Poor flow characteristics
- Reduced mechanical properties
- Increased tool wear
- Surface appearance problems, (especially with blends).

The inclusion of a radius will give:
- Uniform cooling
- Less warpage
- Less flow resistance
- Easier filling
- Lower stress concentration
- Less notch sensitivity.

The outside corner radius should be equal to the inside radius plus the wall thickness as this will keep a uniform wall thickness and reduce stress concentrations.

For a part with an internal radius half the nominal wall thickness a stress concentration factor of 1.5 is a reasonable assumption. For smaller radii, e.g. 10% of the nominal wall, this factor will increase to 3. Standard tables for stress concentration factors are available and should be consulted for critical applications.

In addition, from a molding view point, it is important is to avoid sharp internal corners. Due to the difference in area/volume-ratio of the polymer at the outside and the inside of the corner, the cooling at the outside is better than the cooling at the inside. As a result the material at the inside shows more shrinkage and so the corner tends to deflect (see Figure 41).

Figure 39 Corner radius.

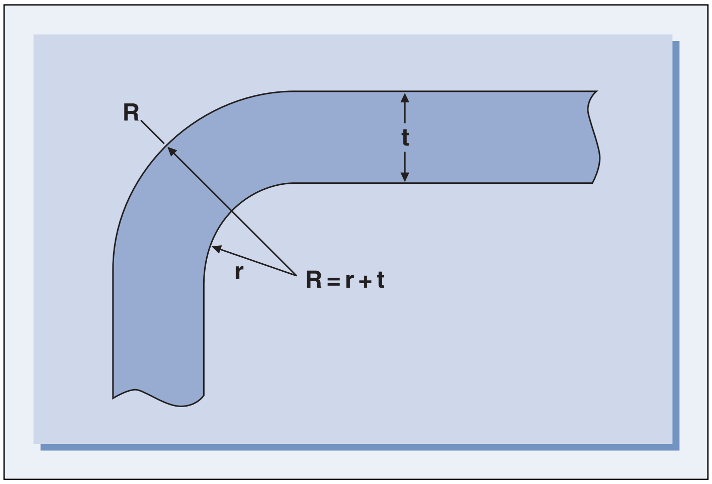


Figure 40 Stress concentration as a function of wall thickness and corner radius.

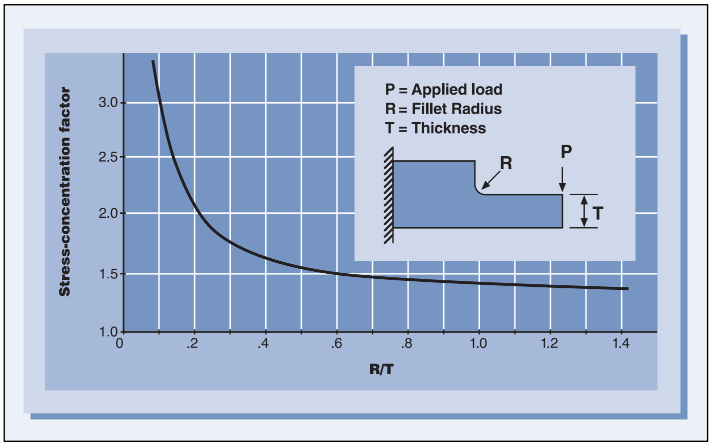
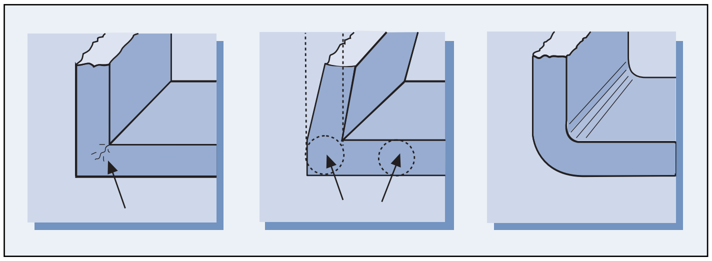


Figure 41 Sharp corners.



Unlimited. DSM

Figure 42 Characteristics of tolerance classes.

Normal	Accurate	Precise
Standard toolmaking techniques	Accurate dimensions mold cavities	High-precision molds
Multiple cavity molds	Multiple cavity molds occasionally	Only single-cavity molds
Conditions adapted for low-cost manufacturing	Molding conditions more critical	Molding conditions carefully controlled
Scrap can be reused	Reuse of injection-molding scrap possible to a limited extent	Process scrap not allowed
Random inspection	Statistical quality control	Statistical process control

Figure 43 Factors affecting parts tolerance.

Part Design	Material properties	Processing	Mold design
Product use	Shrinkage (isotropic	Machine capacity	Mold cavity tolerances
Wall thickness(es)	or anisotropic	Injection pressure/speed	Number of cavities
Draft	Dimensional stability	Holding pressure/time	Runner system
Symmetry	Viscosity	Melt and mold temperature	Ejector system
Surface finish	Reinforcements	Clamping force	Cooling system
Dimensions		Reproducibility	Design/Layout

Tolerances

Establishing the correct tolerances with respect to the product function is of economic importance. The designer should be aware that dimensions with tight tolerances have a big influence on the costs of both product and mold. Even slightly over specifying tolerances may adversely influence tool costs, injection molding conditions, and cycle time. It is recommended to indicate only critical dimensions with tolerances on a drawing. Depending on the application, a division into three tolerance classes can be made:

– normal; price index 100
– accurate; technical injection molding; price index 170
– precise; precision injection molding; price index 300

The most important characteristics of the tolerance classes are shown in Figure 42.

Mold design, mold cavity dimensions, product shape, injection-molding conditions and material properties determine the tolerances that can be obtained. Figure 43 provides a summary of the factors that play a major role in establishing dimensional accuracy.

Coring

Coring refers to the elimination of plastic material in oversized dimensioned areas by adding steel to the mold tool that usually results in a pocket or opening in the part. For simplicity and economical reasons cores should ideally be placed parallel to the line of draw.

Cores in other directions require the use of some form of side action (cam operated or hydraulic) to be actuated thus increasing tooling costs (see Figure 45).

Undercuts

Undercuts should be avoided if possible through redesign of the part. Ideally, the mold tool should open in a direction parallel to the movement of the machine platen. In Figure 46 hanging the form of the hole reduces initial costs and also maintenance cost during production. For some complex parts the ideal situation will not exist and mechanical movement of one sort or another will be required. A description of possible movements includes the following:

- **Deflection –** Dependent on the material and amount of undercut it may be possible to deflect the part out of the mold.

- **Inserts –** The use of removable inserts that eject with the part is an option, certainly for prototype tooling. The disadvantages are the inserts must be removed from the ejected part and repositioned in the mold thus possibly extending the cycle time.

- **Cams –** Cams or hydraulic/pneumatic cylinders move part of the mold out of the way to permit part ejection. These increase the complexity of the mold making it more expensive and also mean a controller is required to operate them during the molding cycle. Cycle times will also be affected.

- **Slides –** By means of angled pins and rods mounted in the mold it may be possible to move the part of the mold forming the undercut in the direction of the angled pin during the opening sequence of the mold. This then allows ejection of the part.

- **Stepped parting line –** By repositioning the parting line it may be possible to eliminate undercut features; although this may add to the complexity of the tool it is the most recommended solution.

Figure 44 Coring designs.

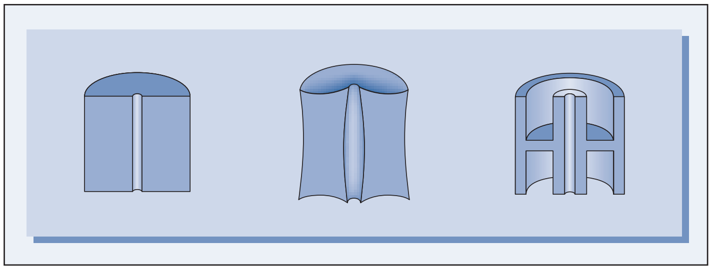


FIgure 45 Construction B could cost as much as 60% less than A.

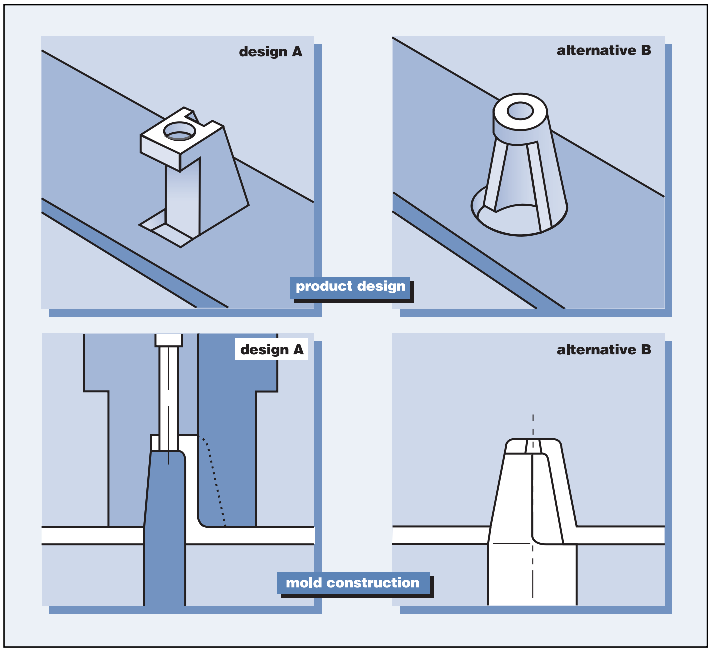
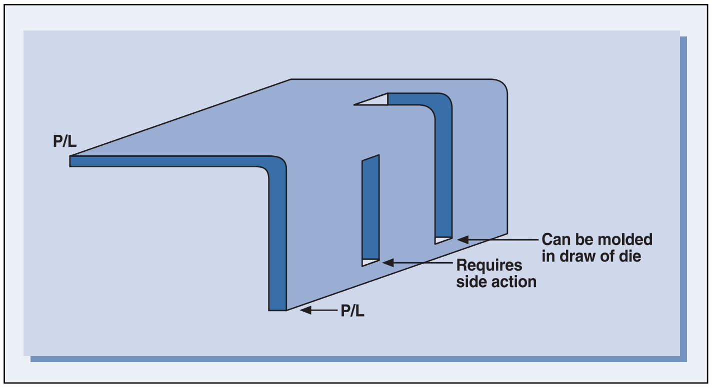


Figure 46 Undercuts.



Unlimited. DSM

Figure 47 Cammed mold for part with undercut cams move in vertical direction when mold is opened.

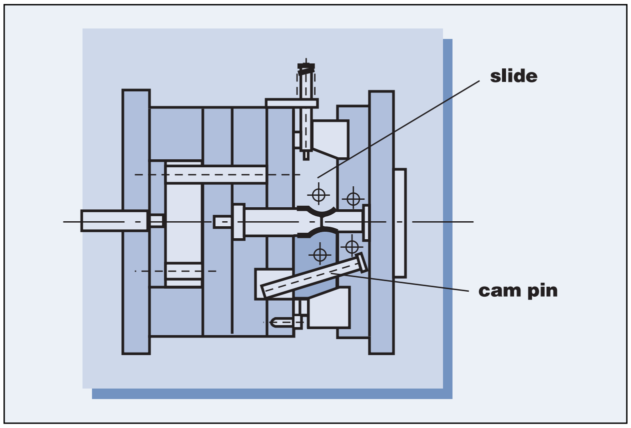


Figure 48 Draft (A) in mm for various draft angles (B) as a function of molding depth (C).

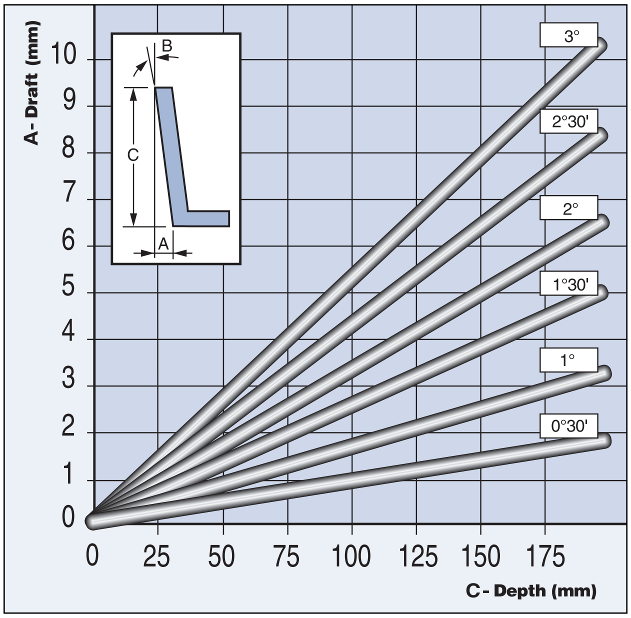


Figure 49 Parting line.

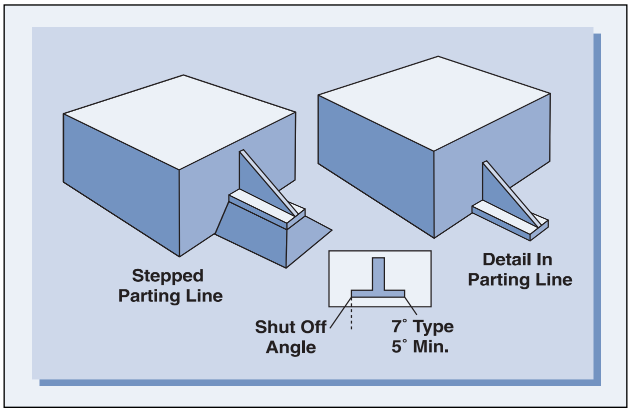


Figure 47 shows an example of a sliding cam. The cam pins that operate the cams are mounted under a maximum angle of 20° - 25° in the injection side. The angle is limited because of the enormous force that is exerted on these pins during mold opening and closing.

Draft Angle

Part features cut into the surface of the mold perpendicular to the parting line require taper or draft to permit proper ejection. This draft allows the part to break free by creating a clearance as soon as the mold starts to open. Since thermoplastics shrink as they cool they grip to cores or male forms in the mold making normal ejection difficult if draft is not included in the design. If careful consideration is given to the amount of draft and shutoff in the mold it is often possible to eliminate side actions and save on tool and maintenance costs. For untextured surfaces generally a minimum of 0.5 deg draft per side is recommended although there are exceptions when less may be acceptable. Polishing in draw line or using special surface treatments can help achieve this. For textured sidewalls use an additional 0.4 deg draft per 0.1mm depth of texture.

Typically 1 to 3 deg draft is recommended. As the draft increases ejection becomes easier but it increases the risk that some sections may become too heavy. Try to keep features in the parting line or plane. When a stepped parting line is required allow 7 deg for shutoff. 5 deg should be considered as a minimum. Drag at the shutoff will cause wear over time with the risk that flash will form during molding. More frequent maintenance will be required for this type of tooling if flash free parts are to be produced.

Mold Design Guidelines

Mold design and construction requires special attention for optimal product quality and reliable molding.

Mold Machine

The mold should be tuned to the injection molding equipment with respect to mold mounting, injection unit and clamping force. Relevant molding machine data can be found in Figure 50.

The maximum shot weight of the injection unit is the amount of plastic that can be injected per shot. The weight of the molding should not exceed 70% of the maximum shot weight.

The minimum plasticizing capacity depends on the relationship between shot weight and cooling time. For example, molding 300 g in 30 seconds requires a minimum plasticating capacity of 36 kg/hr.

The required clamping force of a molding machine is determined by the cavity pressure during the injection/ holding stage and the projected area of the part in the clamp direction. Various factors affect the molding pressure, e.g. length over thickness ratio of the molded part, injection speed and melt viscosity. Typical injection pressures are 40-50 N/mm^2, resulting in a required clamp force of 0.4-0.5 tons/cm^2. For thin wall moldings the required pressures can be a factor 2 higher resulting in a required clamp force of 1 tons/cm^2. With engineering structural foam molding pressures are significantly lower allowing for less robust mold design or the use of aluminum in place of steel in the mold construction.

Figure 50 Mold mounting dimensions.

Molding Machine	Mold
Minimum/maximum mold height	Mold closed height
Opening stroke	Ejection stroke
Tie bars spacing	Mounting plate dimensions
Mounting holes or grooves	Knockout pattern
Knockout pattern	Locating ring diameter
Nozzle alignment	Length of sprue bushing
Insertion depth of nozzle	Sprue bushing radius
Nozzle radius	Sprue orifice diameter
Nozzle orifice	

Figure 51 Impression of a standard injection mold.

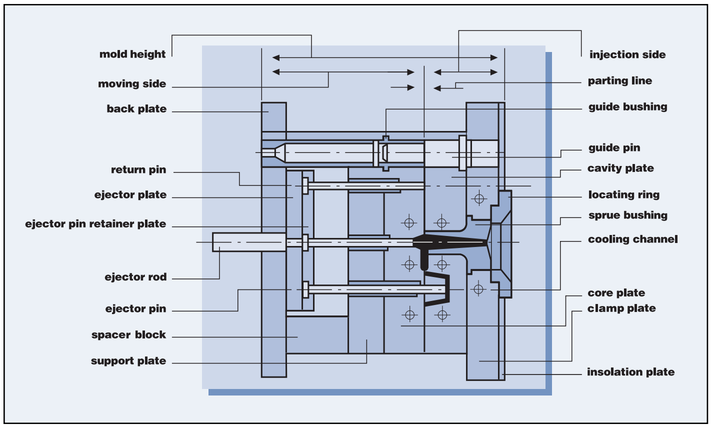
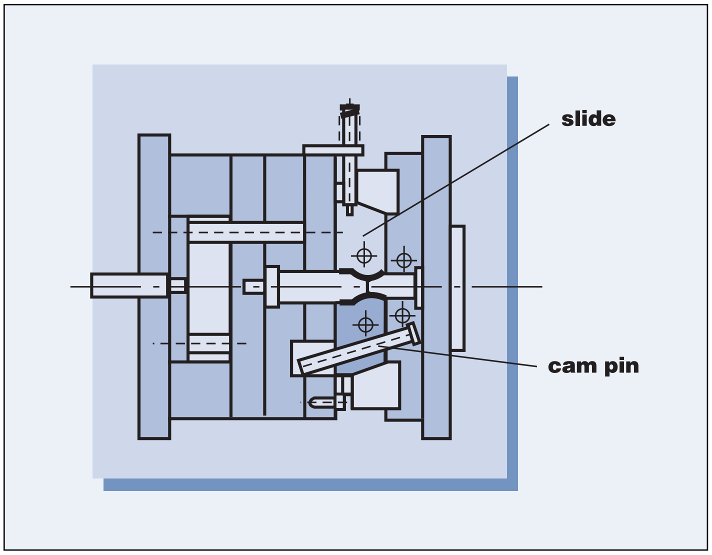


Figure 52 Cammed mold for part with undercut cams move in vertical direction when mold is opened.



Mold Construction

A standard injection mold is made of a stationary or injection side containing one or more cavities and a moving or ejection side. Relevant details are shown in Figure 51.

High quality molds are expensive because labor and numerous high- precision machining operations are time-consuming. Product development and manufacturing costs often can be significantly reduced if sufficient attention is paid to product and mold design. The way in which the mold is constructed is determined by:

- shape of the part
- number of cavities
- position and system of gating
- material viscosity
- mold venting

A simple mold with a single parting line is shown in Figure 51. More complex molds for parts with undercuts or side cores may use several parting lines or sliding cores. These cores may be operated manually, mechanically, hydraulically, pneumatically or electromechanically.

Figure 52 shows an example of a sliding cam. The cam pins that operate the cams are mounted under a maximum angle of 20° - 25° in the injection side. The angle is limited because of the enormous force that is exerted on these pins during mold opening and closing.

Multi-Cavity Molds

The number of cavities and mold construction depend both on economical and technical factors. Important is the number of parts to be molded, the required time, and price in relation to mold manufacturing costs. Figure 53 shows the relation between the total part costs and the number of cavities.

The gating system and gate location can limit the design freedom for multi-cavity molds. Dimensional accuracy and quality requirements should be accounted for. The runner layout of multiple-cavity molds should be designed for simultaneous and even cavity filling. The maximum number of cavities in a mold depends on the total cavity volume including runners in relation to the maximum barrel capacity and clamping force of the injection molding machine.

Number of cavities. A given molding machine has a maximum barrel capacity of 254 cm^3, a plasticizing capacity of 25 g/s, 45 mm screw and a clamping force of 1300 kN. A PC part of 30 cm^3, (shot weight 36 g) and a projected area of 20 cm^2 including runners requires about 0.5/tons/cm^2 (5 kN/cm^2) clamping force.

The maximum number of cavities based on the clamping force would be 12. It is advisable to use only 80% of the barrel capacity, thus the number of cavities in this example is limited to 6.

When very short cycle times are expected the total number of cavities may be further reduced. A 6-cavity mold in this example requires a shot weight of 216 g. The cooling time must be at least 8.7 seconds.

Figure 53 Total part costs in relation to number of cavities.

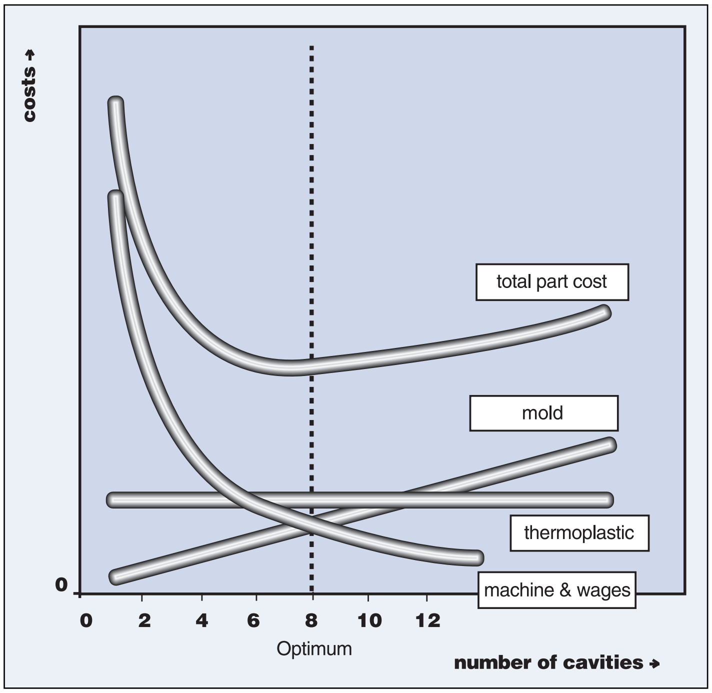


Figure 54 Sprue design.

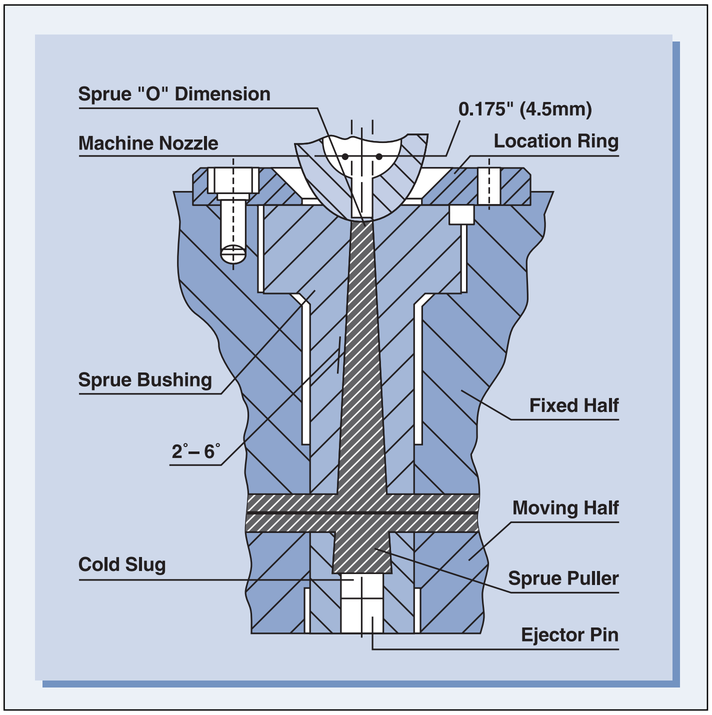
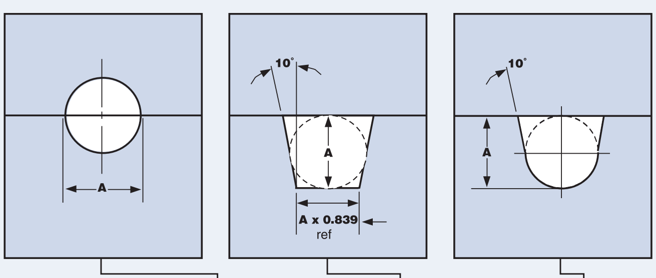


Figure 55 Cross sectional properties for various runner profiles.



A		Full Round		Modified Trapezoid		Modified Trapezoid	
(in)	*(mm)*	*(in2)*	*(mm2)*	*(in2)*	*(mm2)*	*(in2)*	*(mm2)*
1/8	3.2	0.012	8.0	0.016	10.4	0.015	9.7
3/16	4.8	0.028	18.1	0.036	23.4	0.033	21.8
1/4	6.4	0.049	32.2	0.063	41.6	0.059	38.7
5/16	7.9	0.077	49.0	0.099	63.4	0.092	59.0
3/8	9.5	0.110	70.9	0.143	91.6	0.133	85.3
7/16	11.1	0.150	96.8	0.194	125.1	0.181	116.4
1/2	12.7	0.196	126.7	0.254	163.8	0.236	152.4
5/8	15.9	0.307	198.6	0.397	256.7	0.369	238.9

Cold Runner Systems

The runner system is a manifold for distribution of thermoplastic melt from the machine nozzle to the cavities. The sprue bushing and runners should be as short as possible to ensure limited pressure losses in the mold. Properly sizing a runner to a given part and mold design has a tremendous pay-off.

To summarize a runner system that has been designed correctly will:

- Deliver melt to the cavities
- Balance filling of multiple cavities
- Balance filling of multi-gate cavities
- Minimize scrap
- Eject easily
- Maximize efficiency in energy consumption
- Control the filling/packing/cycle time.

Sprue bushings. Sprue bushing connects the machine nozzle to the runner system. To ensure clean ejection from the bushing the bushing should have a smooth, tapered internal finish and have been polished in the direction of draw. The use of a positive sprue puller is recommended. A cold slug well should also be included in the design. This prevents a slug of cold material from entering the feed system and finally the part, which could affect the final properties of the finished part. The dimensions of the sprue depend primarily on the dimensions of the molded part in particular the wall thickness. As a guideline:

- The sprue must not freeze before any other cross section in order to permit sufficient transmission of holding pressure.
- The sprue must de- easily and reliably.

Runner geometry. The ideal runner cross section is circular as this ensures favorable melt flow and cooling. However, it takes more effort to build circular runners because one half must be machined in the fixed mold part and the other half in the moving mold part. The higher the surface to volume ratio, the more efficient the runner.

Full trapezoidal channels in one of the two mold halves provide a cheaper alternative (see Figure 55). The rounded off trapezoidal cross section combines ease of machining in one mold half with a cross section that approaches the desired circular shape. The height of a trapezoidal runner must be at least 80% of the largest width. Half round runners are not recommended because of their low volume to surface ratio.

Runner dimensions. The diameter of a runner highly depends on its length in addition to the part volume, part flow length, machine capacity, and gate size. Generally they must never be smaller than the largest wall thickness of the product and usually lie within the range 3 mm to 15 mm. Recommended runner dimensions are provided in Figure 56. The selection of a cold runner diameter should be based on standard machine tool cutter sizes

Runner layout. There are 3 basic layout systems used for multi-cavity systems. These can be categorized as follows:

- Standard (herringbone) runner system
- "H" bridge (branching) runner system
- Radial (star) runner system

Unbalanced runner systems lead to unequal filling, post-filling and cooling of individual cavities that may cause failures like:

- Incomplete filling
- Differences in product properties
- Shrinkage differences/warpage
- Sink marks
- Flash
- Poor mold release
- Inconsistency

Although the herringbone is naturally unbalanced, it can accommodate more cavities than its naturally balanced counterparts, with minimum runner volume and less tooling cost. With computer aided flow simulation it is possible to adjust primary and secondary runner dimensions to obtain equal filling patterns.

Figure 56 Maximum runner length for specific diameters.

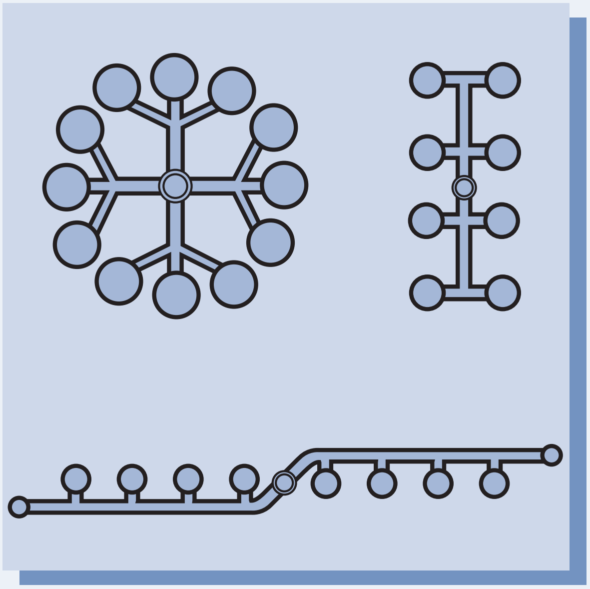
Runner Diameter		Maximum Runner Length			
		Low Viscosity		High Viscosity	
(in)	(mm)	(in)	(mm)	(in)	(mm)
1/8	3	4	100	2	50
1/4	6	8	200	4	100
3/8	9	11	280	6	150
1/2	13	13	330	7	175

$$D = \frac{w^{1/2} \times L^{1/4}}{3.7}$$

where:

D = runner diameter (mm)
W = part weight (g)
L = runner length (mm)

Figure 57 Example of unbalanced feed systems.



Unlimited. DSM

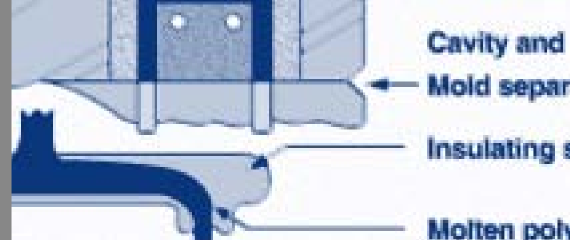


Figure 58 Naturally balanced feed systems.

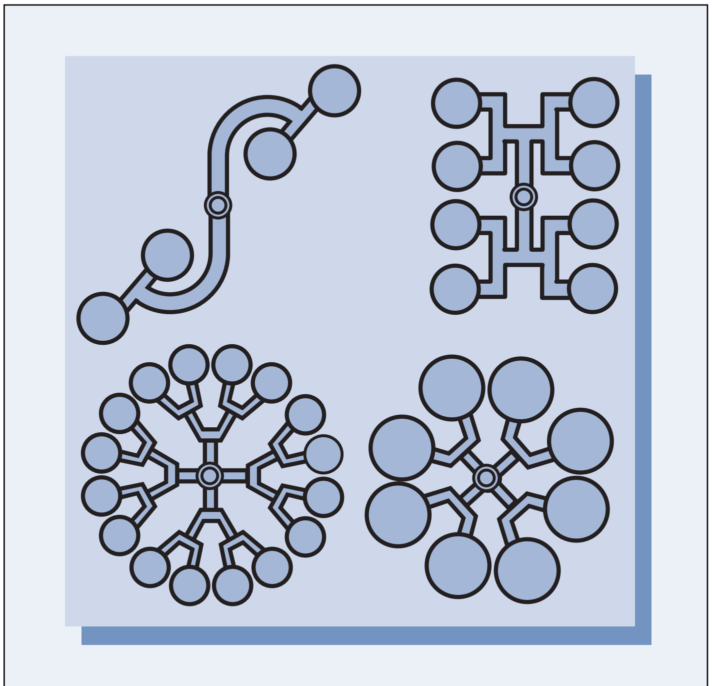
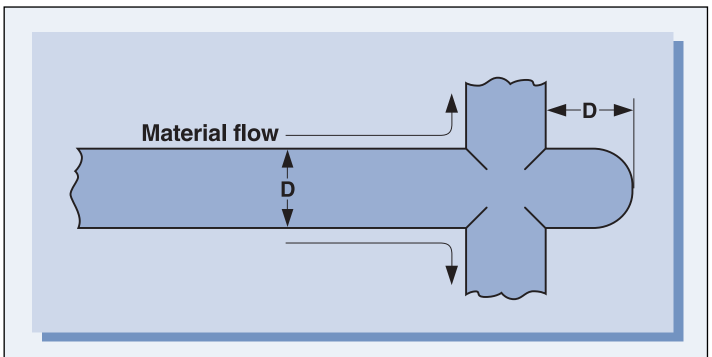


Figure 59 Recommended design of cold slug well or overflow.



Keep in mind that non-standard runner diameters will increase manufacturing and maintenance costs. Adjusting runner dimensions to achieve equal filling may not be sufficient in critical parts to prevent potential failures. Special attention is required for:

- Very small components
- Parts with thin sections
- Parts that permit no sink marks
- Parts with a primary runner length much larger than secondary runner length.

It is preferred to design naturally balanced runners as shown in Figure 58.

The "H" (branching) and radial (star) systems are considered to be naturally balanced. The naturally balanced runner provides equal distance and runner size from the sprue to all the cavities, so that each cavity fills under the same conditions. When high quality and tight tolerances are required the cavities must be uniform.

Family molds are not considered suitable. Nevertheless, it might be necessary for economical reasons to mold different parts in one mold. The cavity with the largest component should be placed nearest to the sprue. The maximum number of cavities in a mold depends on the total cavity volume including runners in relation to the maximum barrel capacity and clamping force of the injection molding machine.

Branched runners. Each time a runner is branched, the diameter of the branch runners should be smaller than the main runner, because less material flows through the branches and it is economically desirable to use minimum material in the runners. Where N is the number of branches, the relationship between the main runner diameter (dmain) and the branch runner diameter (dbranch) is:

dmain= dbranch x N1/3

At all runner intersections there should be a cold slug well. The cold slug well helps the flow of material through the runner system by stopping colder, higher viscosity material moving at the forefront of the molten mass entering into the cavity.

The length of the well is usually equal to or greater than the runner diameter and this is achieved by extending the length of the primary runner at the intersection with the secondary runner (see Figure 59).

While large runners facilitate the flow of material at relatively low pressure requirements, they require a longer cooling time, more material consumption and scrap, and more clamping force. Designing the smallest adequate runner system will maximize efficiency in both raw material use and energy consumption in molding. The runner size reduction is constrained by the molding machine's injection pressure capability.

Initially runner diameters can be calculated with the following formula below. Further fine-tuning can then be performed with the use of flow analysis software where effects such as shear heating and skin layer formation can be taken into consideration.

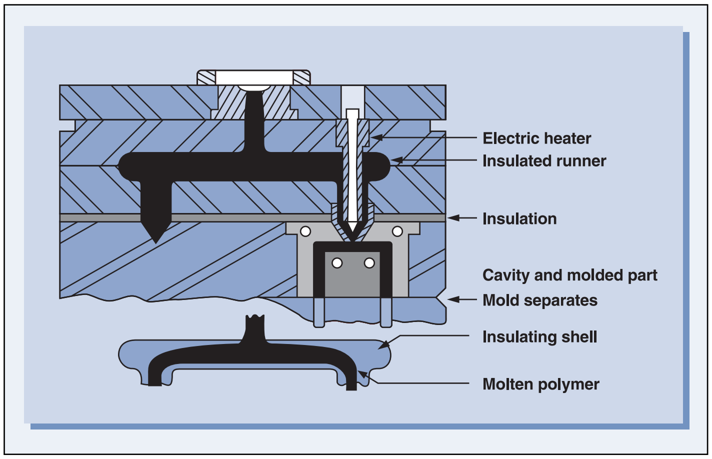
Hot Manifold / Runnerless Molds

Runnerless molds differ from cold runner molds by extending the injection molding machine's melt chamber and acting as an extension of the machine nozzle. A portion or all of the polymer melt is at the same temperature and viscosity as the polymer in the barrel of the injection molding machine. There are two general types of runnerless molds – the insulated system and the hot runner system.

Insulated runner system. Insulated runner molds have oversized passages formed in the mold plate. The passages are of sufficient size that, under conditions of operation, the insulated effect of the plastic (frozen on the runner wall) combined with the heat applied with each shot maintains an open, molten flow path.

The insulated runner system should be designed so that while the runner volume does not exceed the cavity volume all of the molten material in the runner is injected into the cavity during each shot. This helps prevent excessive build-up of the insulating skin and minimizes any drop in melt temperature.

Figure 60 Insulated runner mold.



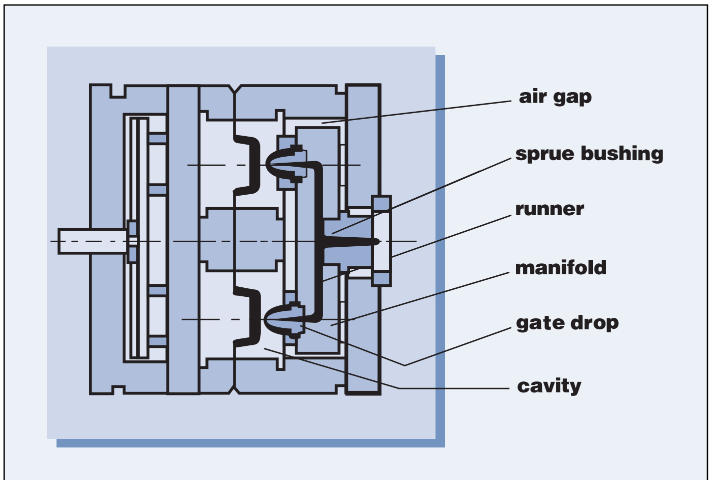
Advantages of an insulated runner system compared to a conventional cold runner system include:

- Reduction in material shear
- Faster cycle times
- Elimination of runner scrap
- Decreased tool wear
- Improved part finish
- Less sensitivity to the requirements for balanced runners
- Shorter cycle times

Disadvantages include:

- (Generally) more complex tool design
- (Generally) higher tool costs
- Higher maintenance costs
- Start up procedure is more difficult
- Possible thermal degradation of material
- More difficult to change color

Figure 61 Cross section of a basic hot runner system.



Hot runners. More commonly used than insulated runners are hot runners which fall into two categories: internally and externally heated.

Hot runners retain the advantages of the insulated runner system over conventional cold runner system and eliminate a number of the disadvantages. The major disadvantages compared to a cold runner mold are:

- More complex mold design, manufacture operation and maintenance
- Substantially higher costs
- Thermal expansion of various components needs to be taken into account

These disadvantages are a result of the need to install a heated manifold, balance heat generated by the manifold and the minimization of polymer hang-ups. Figure 61 shows a schematic cross-section of a hot runner system. It is often cost effective to produce large volumes with hot runner molds, in spite of high investments. These systems are used for a wide range of applications.

The electrical/electronic industry uses small components, like connectors and bobbins that are molded in multi-cavity molds. On the other hand, large multi-gated parts are used in the automotive industry, e.g. bumpers and dashboards. Yet both can benefit from the cost and technical advantages of hot runners.

Cycle time reduction is possible when cooling of a cold runner would determine the cycle time. Following are typical advantages and disadvantages of hot runner systems:

Advantages

- Production increase (cycle)
- Material saving
- Quality improvement
- No waste
- Automatic degating
- Energy saving
- Flexible choice gate location

Disadvantages

- Higher investment
- Critical molding conditions
- Critical temperature control
- Start-up problems (tailing)
- Color change problems
- Abrasion (reinforced plastics)
- Critical mold design (no dead spots)

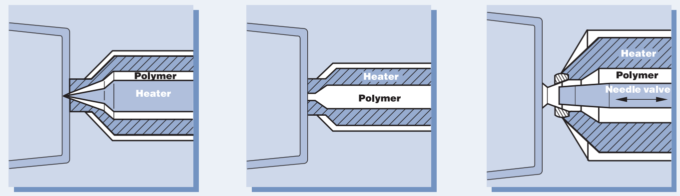
In selecting a hot runner system, the factors in Table 1 need to be taken into account. Taking all these factors into consideration, there is still a choice between many types and variations of hot runner manifolds and nozzles. General recommendations cannot be given. The best option depends on the thermoplastic and the requirements of the specific application.

The following guidelines should be respected:

- Natural runner balancing
- Minimal pressure-losses
- Sufficient heating capacity for manifold and each single nozzle
- Accurate, separate temperature controls for manifold and nozzle
- Effective insulation between manifold and mold
- Optimal mold temperature control
- No dead spots and flow restrictions in manifold and nozzles
- Limited residence time of melt in the hot runner
- Adequate sealing of runners

Figure 62 shows various basic types of nozzle configurations with their typical advantages and disadvantages. With respect to externally and internally heated manifolds the same conclusions are applicable as for nozzles. A relatively cheap and robust alternative for hot runners is the hot runner/ cold sprue. The hot runner manifold is followed by a short cold sprue that eliminates the use of expensive nozzles.

Figure 62 Advantages and disadvantages of basic hot runner configurations.



Internally heated nozzle/manifold.	Externally heated nozzle/manifold.	Externally heated nozzle with needle valve.
Advantage - simple and cheap - solid construction - lower energy cost - no leakage	Advantage - homogeneous melt temperature - small melt-volume - applicable for broad range of materials - low pressure-loss - easy color change	Advantage - homogeneous melt temperature - small melt-volume - large gate diameter possible - applicable for high viscosity and thermally instable materials - process control
Disadvantage - temperature homogeneity - mainly for polyolefines - higher pressure losses - difficult color changes - high shear materials - process control	Disadvantage - high energy input - possible leakage - insulation from cold mold parts necessary - costs	Disadvantage - high energy input - insulation from cold mold necessary - costs - high maintenance - high shear

Table 1 Factors to consider in selecting a hot runner system.

Economy	**Product**
Investment	Dimensions
Number of parts	Shot weight
Cycle times	Gate/sink marks
Material waste	Reproducibility
Energy savings	Required tolerances/warpage
Regrinding	Fiber orientation
Process	**Material**
Start up	Flow behavior
Total flow path	Melting temperature/range
Pressure distribution	Process window
Melt homogeneity	Thermal stability
Color change	Reinforcement
Residence time	Additives

Unlimited. DSM

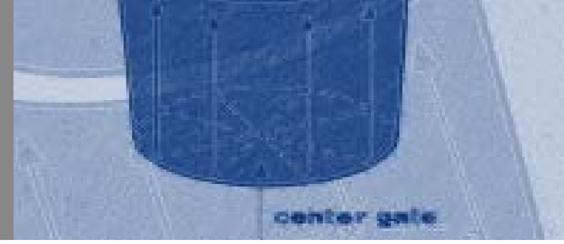
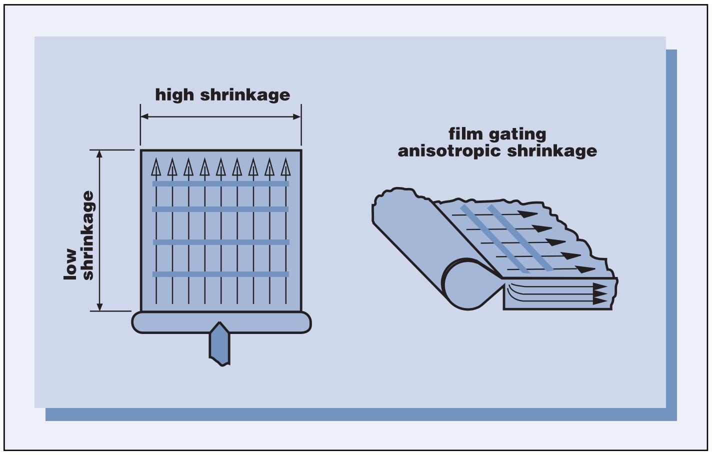


Figure 63 Influence of gating on glass fiber orientation and shrinkage of the product.



Gate Design

Gates are a transition zone between the runner system and the cavity. The location of gates is of great importance for the properties and appearance of the finished part. The melt should fill the entire cavity quickly and evenly. For gate design the following points should be considered:

- Locate the gate at the thickest section
- Note gate marks for aesthetic reasons
- Avoid jetting by modifying gate dimensions or position
- Balance flow paths to ensure uniform filling and packing
- Prevent weld lines or direct to less critical sections
- Minimize entrapped air to eliminate burn marks
- Avoid areas subject to impact or mechanical stress
- Place for ease of degating

Single vs. multiple gates. Unless the length of the melt flow exceeds practical limits a single gate is the preferred option. Multiple gates always create weld lines where the flows from the separate gates meet.

A distinction can be made between center and edge gating of a part. Center gated parts show a radial flow of the melt. This type of gate is particularly good for symmetrical parts, such as cup shaped products or gears, because it will assure more uniform distribution of material, temperatures, and packing, and better orientation effects it gives very predictable results. On the other hand, linear flow and cross flow properties often differ. In flat parts, this can induce additional stress and results in warpage or uneven shrinkage.

Because of their simplicity and ease of manufacture, edge gates are the most commonly used. These work well for a wide variety of parts that are injection molded. Long narrow parts typically use edge gates at or near one end in order to reduce warpage.

But it is very difficult to mold round parts using this type gate, as they tend to warp into an oval shape. While a single gate into the body of the part might incur a higher initial tool cost, lower scrap rates and higher part quality will quickly justify this expense.

Gate dimensions. The cross section of the gate is typically smaller than that of the part runner and the part, so that the part can easily be "de-gated" (separated from the runner) without leaving a visible scar on the part. The gate thickness is typically between one half and two-thirds the part thickness. Since the end of packing can be identified as the time when the material in the gate drops below the freeze temperature, the gate thickness controls the packing time.

A larger gate will reduce frictional heating, permit lower velocities, and allow the application of higher packing pressure for a longer period of time. If appearance, low residual stress, and better dimensional stability were required then a larger gate would be advantageous.

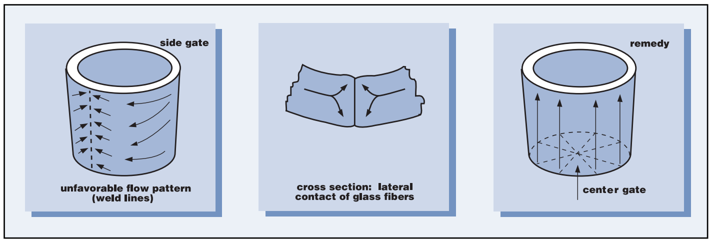
A minimum size of 0.8 mm is recommended for unreinforced materials. Smaller gates may induce high shear and thus thermal degradation. Reinforced thermoplastics require slightly larger gates > 1 mm. As a rule it should not exceed the runner or sprue diameter. The maximal land length should be 1 mm.

Gate location. Location items to consider include:

Appearance
Whenever possible locate gates on non-visual surfaces thus eliminating problems with residual gate vestiges after the gate has been removed.

Stress
Avoid areas exposed to high external stress (mechanical or impact). The gate area has high residual stresses and also rough surfaces left by the gate act as stress concentrators.

Figure 64 Influence of gate location on flow behavior of the melt.



Pressure
Locate the gate in the thickest section to ensure adequate pressure for packing out the part. This will also help prevent sink marks and voids forming.

Orientation
Molecular orientation becomes more pronounced in thin sections, the molecules usually align themselves in the flow direction. High degrees of orientation result in parts having unaixal strength, resistance to loading only in one direction. To minimize molecular orientation the gate should be located so that as the melt enters the cavity it is diverted by an obstruction such as the cavity wall or an ejection pin.

Weld lines
Place gates to minimize the number and length of weld lines or to direct weld lines to positions that are not objectionable to the function or appearance of the part. When weld lines are unavoidable try to locate the gates close to the weld line location this should help maintain a high melt temperature that is beneficial to a strong weld line.

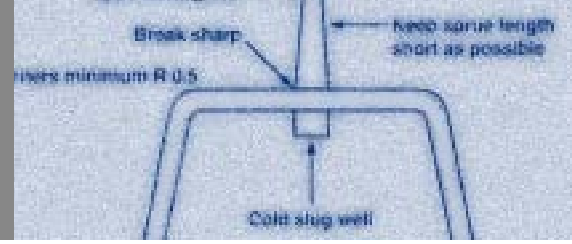
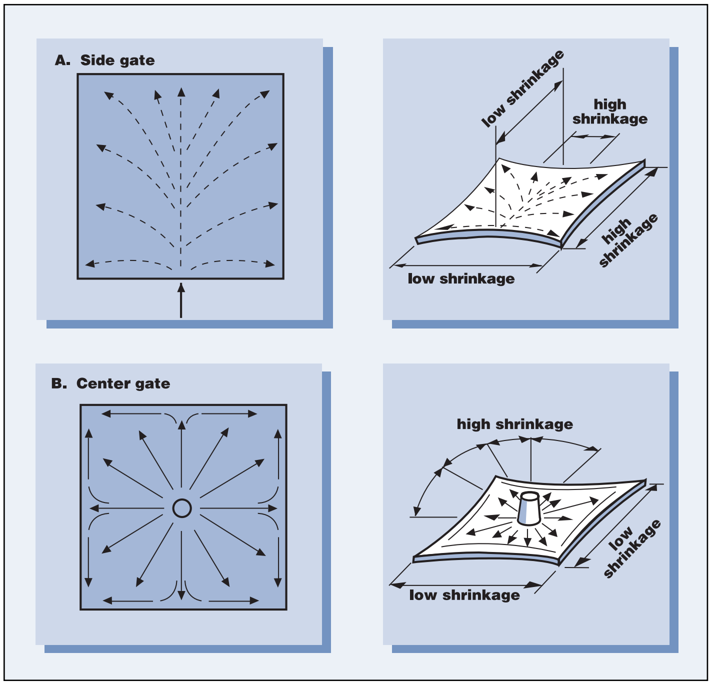


Figure 65 Warpage due to unfavorable gate location.



Glass fiber reinforced materials
Fiber-filled materials require larger gates to minimize breakage of the fibers when they pass through the gate. Using small gates such as submarine, tunnel, or pin gates can damage the fillers in filled materials. Gates that deliver a uniform filling pattern (such as an edge gate) and thus, a uniform fiber orientation distribution are preferable to point-type gates. Fiber orientation will normally be the determining factor for warpage problems with this type of material and the gate location and choice of gate type are 2 of the primary factors in controlling the orientation.

In general, there will be a higher glass fiber orientation in thinner wall sections, e.g. less than 2 mm and as injection speed increases. A high injection speed is required to obtain a smooth surface. The direction of orientation is influenced by gate type and location and, of course, by the shape of the product (see Figure 63).

Warpage
An incorrectly dimensioned or located gate may also result in undesirable flow patterns in the cavity. This can lead to moldings with visible weld line (see Figure 64). Undesirable flow patterns in the cavity can also lead to deformation by warping or bending (see Figure 65).

Manually trimmed gates. Manually trimmed gates are those that require an operator to separate parts from runners during a secondary operation. The reasons for using manually trimmed gates are:

- The gate is too bulky to be sheared from the part as the tool is opened.
- Some shear-sensitive materials (e.g., PVC) should not be exposed to the high shear rates inherent to the design of automatically trimmed gates.
- Simultaneous flow distribution across a wide front to achieve specific orientation of fibers of molecules often precludes automatic gate trimming.

Gate types trimmed from the cavity manually include sprue gates, edge gates, tab gates, overlap gates, fan gates, film gates, diaphragm gates, external rings, and spoke or multipoint gates.

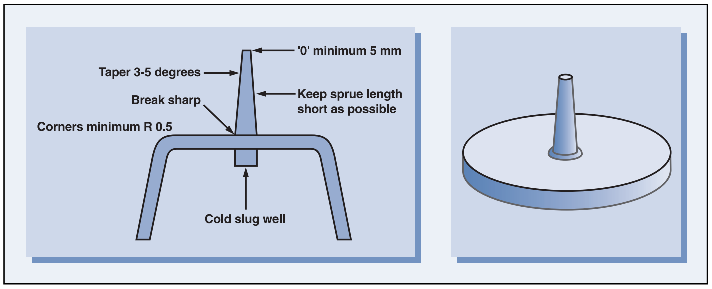
Sprue Gate

Recommended for single cavity molds or for parts requiring symmetrical filling. This type of gate is suitable for thick sections because holding pressure is more effective. A short sprue is favored, enabling rapid mold filling and low-pressure losses. A cold slug well should be included opposite the gate. The disadvantage of using this type of gate is the gate mark left on the part surface after the runner (or sprue) is trimmed off. Freeze-off is controlled by the part thickness rather than determined the gate thickness. Typically, the part shrinkage near the sprue gate will be low; shrinkage in the sprue gate will be high. This results in high tensile stresses near the gate.

The starting sprue diameter is controlled by the machine nozzle. The sprue diameter here must be about 0.5 mm larger than the nozzle exit diameter. Standard sprue bushings have a taper of 2.4 degrees, opening toward the part. Therefore, the sprue length will control the diameter of the gate where it meets the part; the diameter should be at least 1.5 mm larger than or approximately twice the thickness of the part at that point. The junction of sprue and part should be radiused to prevent stress cracking.

- A smaller taper angle (a minimum of one degree) risks not releasing the sprue from the sprue bushing on ejection.
- A larger taper wastes material and extends cooling time.
- Non-standard sprue tapers will be more expensive, with little gain.

Figure 66 Sprue gate.



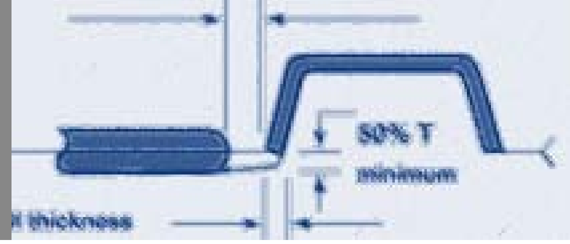


Figure 67 Edge gate.

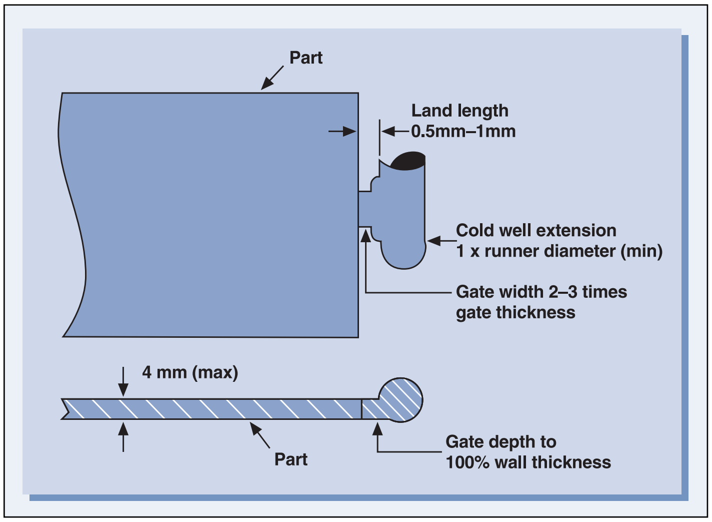


Figure 68 Tab gate.

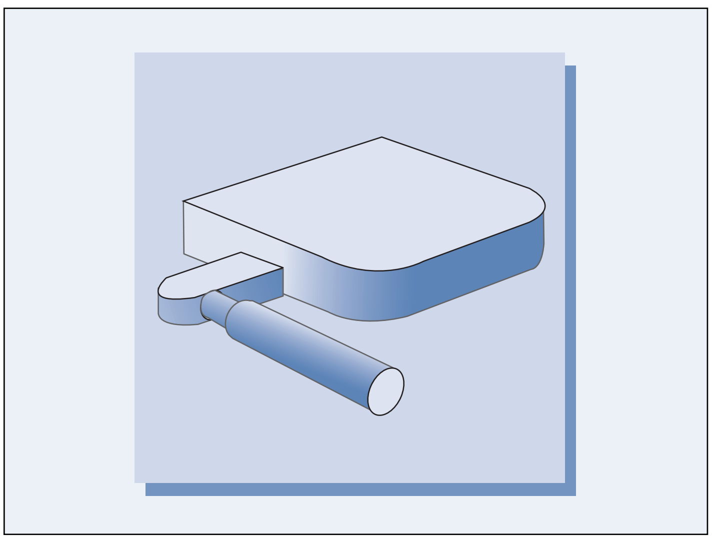
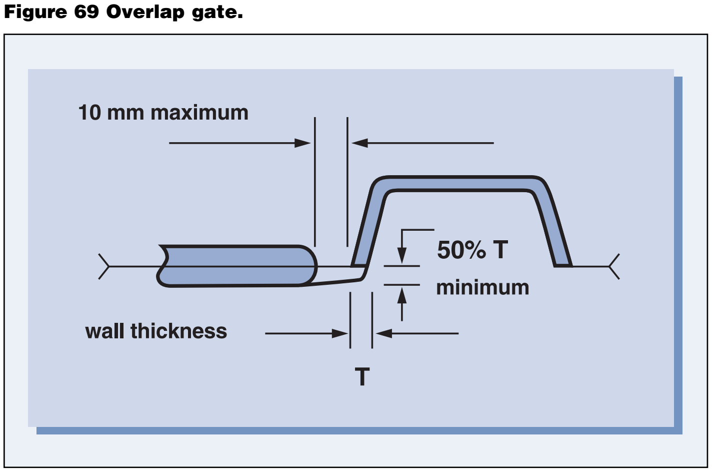


Figure 69 Overlap gate.



Edge Gate

The edge or side gate is suitable for medium and thick sections and can be used on multi-cavity two plate tools. The gate is located on the parting line and the part fills from the side, top or bottom. The typical gate size is 80% to 100% of the part thickness up to 3.5 mm and 1.0 to 12 mm wide. The gate land should be no more than 1.0 mm in length, with 0.5 mm being the optimum.

Tab Gate

A tab gate is typically employed for flat and thin parts, to reduce the shear stress in the cavity. The high shear stress generated around the gate is confined to the auxiliary tab, which is trimmed off after molding. A tab gate is often used for molding P. The minimum tab width is 6 mm. The minimum tab thickness is 75% of the depth of the cavity.

Overlap Gate

An overlap gate is similar to an edge gate, except the gate overlaps the wall or surfaces. This type of gate is typically used to eliminate jetting. The typical gate size is 10% to 80% of the part thickness and 1.0 to 12 mm wide. The gate land should be no more than 1.0 mm in length, with 0.5 mm being the optimum.

Fan Gate

A fan gate is a wide edge gate with variable thickness. This type is often used for thick-sectioned moldings and enables slow injection without freeze-off, which is favored for low stress moldings or where warpage and dimensional stability are main concerns. The gate should taper in both width and thickness, to maintain a constant cross sectional area. This will ensure that:

– The melt velocity will be constant.
– The entire width is being used for the flow.
– The pressure is the same across the entire width.

As with other manually trimmed gates, the maximum thickness should be no more than 80% of the part thickness. The gate width varies typically from 6 mm up to 25% of the cavity length.

Film or Flash Gate

A film or flash gate consists of a straight runner and a gate land across either the entire length or a portion of the cavity. It is used for long flat thin walled parts and provides even filling. Shrinkage will be more uniform which is important especially for fiber reinforced thermoplastics and where warpage must be kept to a minimum. The gate size is small, typically 0.25mm to 0.5mm thick. The land area (gate length) must also be kept small, approximately 0.5 to 1.0 mm long.

Diaphragm Gate

A diaphragm gate is often used for gating cylindrical or round parts that have an open inside diameter. It is used for single cavity molds that have a small to medium internal diameter. It is used when concentricity is important and the presence of a weld line is not acceptable. Typical gate thickness is 0.25 to 1.5 mm.

External Ring Gate

This gate is used for cylindrical or round parts in a multicavity mold or when a diaphragm gate is not practical. Material enters the external ring from one side forming a weld line on the opposite side of the runner this weld line is not typically transferred to the part. Typical gate thickness is 0.25 to 1.5 mm.

Figure 70 Film or flash gate.

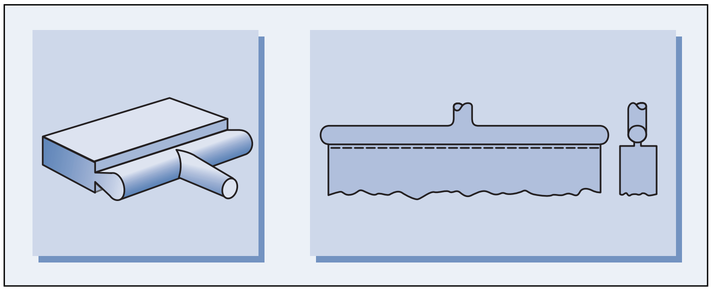


Figure 71 Internal ring gate.

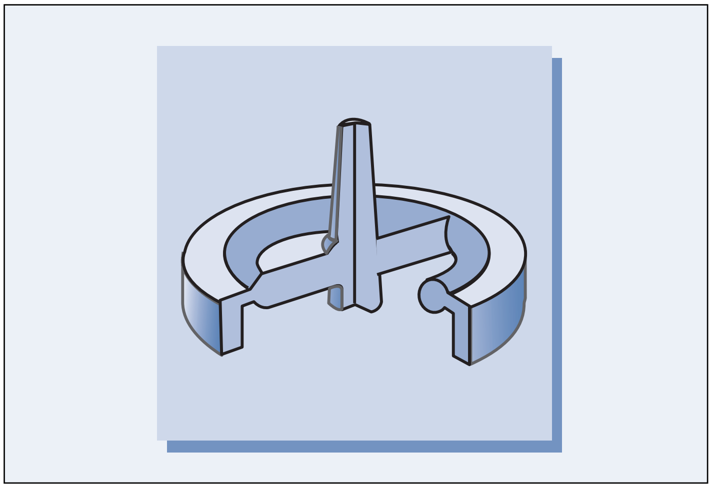
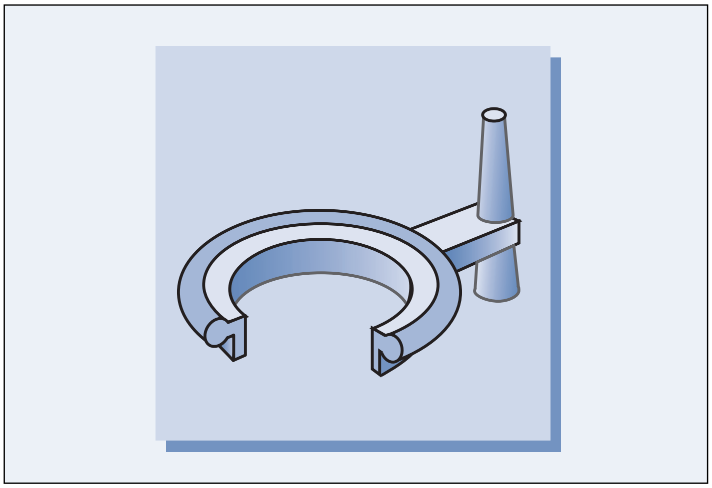


Figure 72 External ring gate.



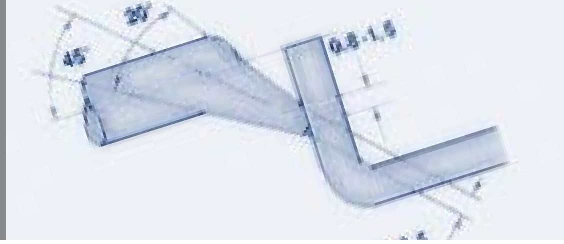


Figure 73 Multi-point gate.

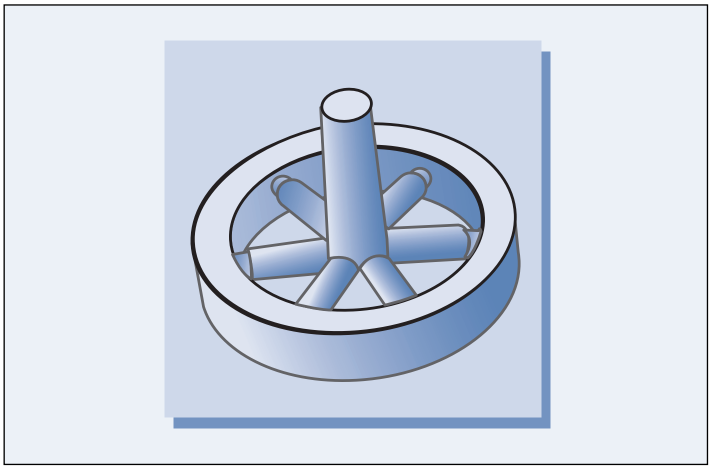


Figure 74 Pin gates.

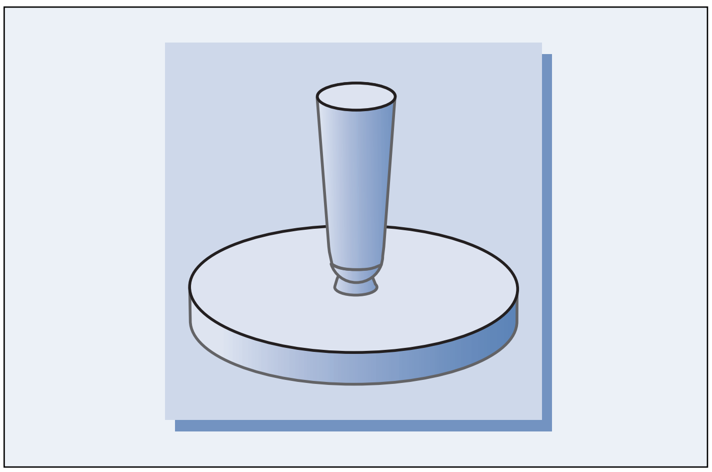


Figure 75 Dimensions of gates (* wall thickness larger than 5 mm should be avoided).

Wall Thickness mm (in)	Gate Diameter / Length mm (in)
0.7 - 1.2 mm (0.02 - 0.05)	0.7 - 1.0 / 0.8 - 1 (0.02 - 0.04 / 0.03 - 0.04)
1.2 - 3.0 mm (0.05 - 0.12)	0.8 - 2.0 / 0.8 - 1 (0.03 - 0.08 / 0.03 - 0.04)
3.0 - 5.0 mm (0.12 - 0.20)	1.5 - 3.5 / 0.9 - 1 (0.06 - 0.14 / 0.04 - 0.04)
≥ 5.0* (0.20)	3.5 - 6.0 / 0.9 - 1 (0.14 - 0.24 / 0.03 - 0.04)

Spoke Gate or Multipoint Gate

This kind of gate is used for cylindrical parts and offers easy de-gating and material savings. Disadvantages are the possibility of weld lines and the fact that perfect roundness is unlikely. Typical gate size ranges from 0.8 to 5 mm diameter.

Automatically trimmed gates. Automatically trimmed gates incorporate features in the tool to break or shear the gate as the molding tool is opened to eject the part. Automatically trimmed gates should be used to:

– Avoid gate removal as a secondary operation.
– Maintain consistent cycle times for all shots.
– Minimize gate scars.

Gate types trimmed from the cavity automatically include pin gates, submarine (tunnel) gates, hot runner gates, valve gates.

Pin Gates

Pin gates are only feasible with a 3-plate tool because it must be ejected separately from the part in the opposite direction The gate must be weak enough to break off without damaging the part. This type of gate is most suitable for use with thin sections. The design is particularly useful when multiple gates per part are needed to assure symmetric filling or where long flow paths must be reduced to assure packing to all areas of the part. Gate diameters for unreinforced thermoplastics range from 0.8 up to 6 mm. Smaller gates may induce high shear and thus thermal degradation. Reinforced thermoplastics require slightly larger gates > 1 mm. The maximal land length should be 1 mm. Advised gate dimensions can be found in Figure 74.

Submarine (tunnel) Gates
A submarine gate is used in two-plate mold construction. An angled, tapered tunnel is machined from the end of the runner to the cavity, just below the parting line. As the parts and runners are ejected, the gate is sheared at the part. The tunnel can be located either in the moving mold half or in the fixed half. A sub-gate is often located into the side of an ejector pin on the non-visible side of the part when appearance is important. To degate, the tunnel requires a good taper and must be free to bend. Typical gate sizes 0.8 mm to 1.5 mm, for glass reinforced materials sizes could be larger.

Hot Runner Gates
Hot runner gates are also known as sprueless gating. The nozzle of a runnerless mold is extended forward to the part and the material is injected through a pinpoint gate. The face of the nozzle is part of the cavity surface; this can cause appearance problems (matt appearance and rippled surface). The nozzle diameter should therefore be kept as small as possible. Most suitable for thin walled parts with short cycle times, this avoid freezing of the nozzle.

Valve Gates
The valve gate adds a valve rod to the hot runner gate. The valve can be activated to close the gate just before the material near the gate freezes. This allows a larger gate diameter and smoothes over the gate scar. Since the valve rod controls the packing cycle, better control of the packing cycle is maintained with more consistent quality.

Figure 76 Tunnel gate.

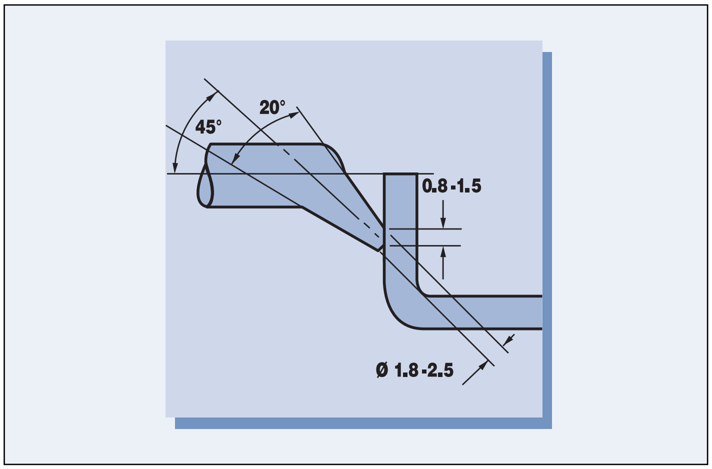


Figure 77 Hot runner gates.

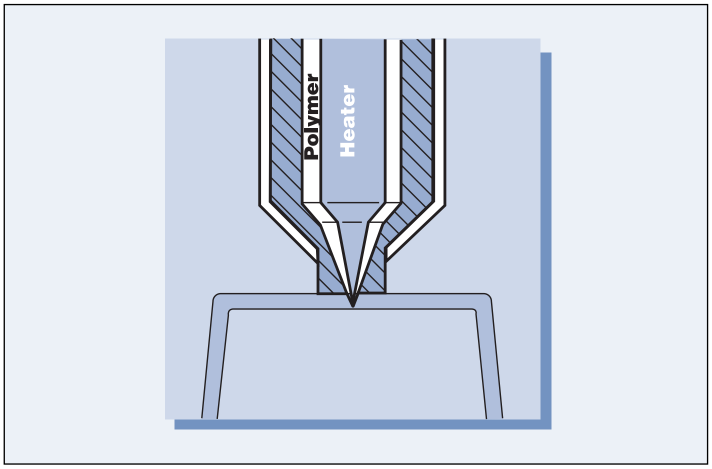
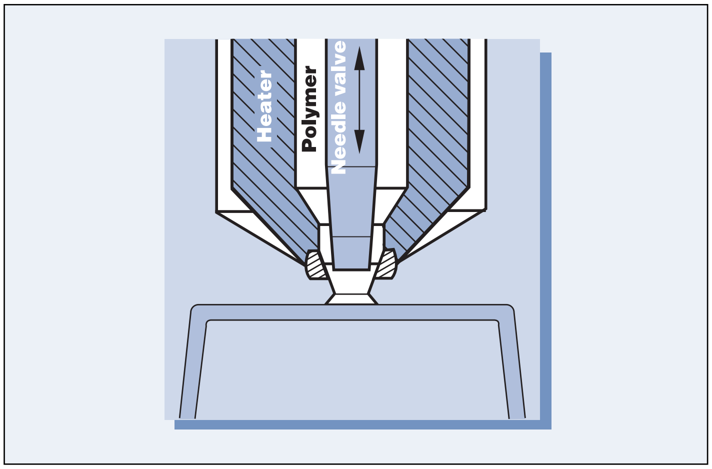


Figure 78 Valve gate.



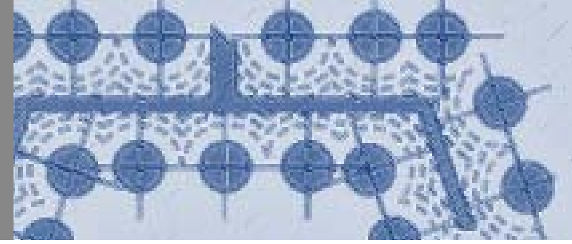


Figure 79 Basic principle of cooling channels.

"w" wall thickness of the product mm (in)	"d" diameter of the cooling channels mm (in)	"a" center distance with respect to mold cavity	"b" center distances between cooling channels
2 (0.08)	8-10 (0.31-0.40)		
2-4 (0.08-0.16)	10-12 (0.40-0.47)	1.5-2d	2-3d
4-6 (0.16-0.24)	12-14 (0.47-0.55)		

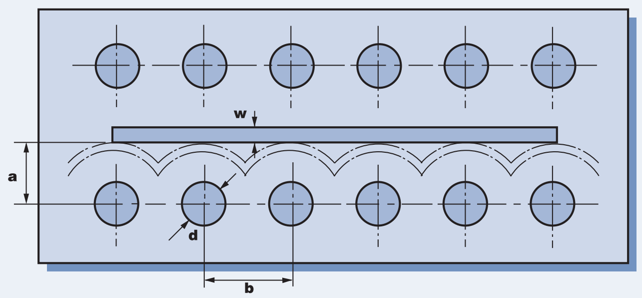
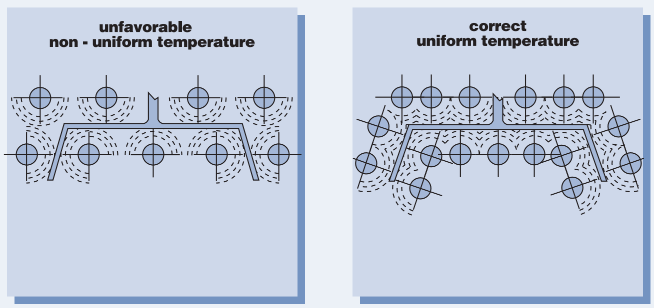


Figure 80 Position of cooling channels.



Mold Cooling

Mold cooling serves to dissipate the heat of the molding quickly and uniformly. Fast cooling is necessary to obtain economical production and uniform cooling is required for product quality. Adequate mold temperature control is essential for consistent molding. The layout of the cooling circuit warrants close attention especially if you consider cooling typically accounts for two thirds of a products cycle time.

Optimal properties of engineering plastics can be achieved only when the right mold temperature is set and maintained during processing. The mold temperature has a substantial effect on:

- Mechanical properties
- Shrinkage behavior
- Warpage
- Surface quality
- Cycle time
- Flow length in thin walled parts

In particular semi-crystalline thermoplastics need to cool down at optimal crystallization rate. Parts with widely varying wall thicknesses are likely to deform because of local differences in the degree of crystallization. Additionally the required cooling time increases rapidly with wall thickness. This calculation is shown in Cooling system equations.

Cooling channel configuration. In general, the cooling system will be roughly drilled or milled. Rough inner surfaces enhance turbulent flow of coolant, thus providing better heat exchange. Turbulent flow achieves 3 to 5 times as much heat transfer as does non turbulent flow. Cooling channels should be placed close to the mold cavity surface with equal center distances in between (see Figures 79 and 80). The mechanical strength of the mold steel should be considered when designing the cooling system.

Some thermoplastics may require mold temperatures of 100°C (212˚F) or higher for optimal processing and properties. Effective mold insulation is advised to minimize heat loss between the mold and the machine mounting platens. Insulation boards with low thermal conductivity and relatively high compressive strength are commercially available.

Care is required in the correct placing of seals; they may be damaged by the sharp edges of the pocket when the mold insert is mounted (see Figure 81). Seals or O-rings should be resistant to elevated temperatures and oils.

Guidelines for optimal mold temperature control include:

- Independent symmetrical cooling circuits around the mold cavities.
- Cores need effective cooling (see baffles, bubblers & thermal pins).
- Short cooling channels to ensure temperature differences between in- and outlet do not exceed 5˚C (41˚F).
- Parallel circuits are preferred over serial cooling as shown in Figure 82.
- Avoid dead spots and/or air bubbles in cooling circuits.
- Heat exchange between mold and machine should be minimized.
- Differences in flow resistance of cooling channels, caused by diameter changes, should be avoided.

Mold parts that are excessively heated, like sprue bushings and areas near the gates, must be cooled intensively. Rapid and even cooling is enhanced by the use of highly conductive metals, such as beryllium-copper. These metals are used to full advantage in places where it is impossible to place sufficient cooling channels. Beryllium copper transfers twice as much heat as carbon steel and four times as much heat as stainless steel. This does not mean beryllium copper molds will run 4 times faster tan a stainless steel mold but they will run some thin walled parts significantly faster. Beryllium copper is not recommended for materials that require high mold temperatures as they allow so much heat transfer that it is difficult to maintain adequate heat economically.

Figure 81 Sealing and cooling channel lay-out.

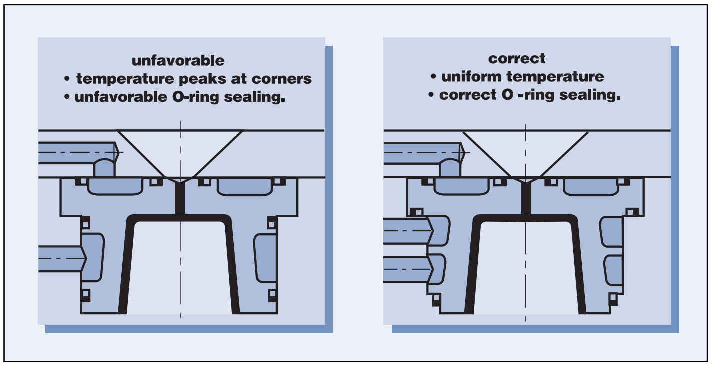


Figure 82 Cooling of the mold.

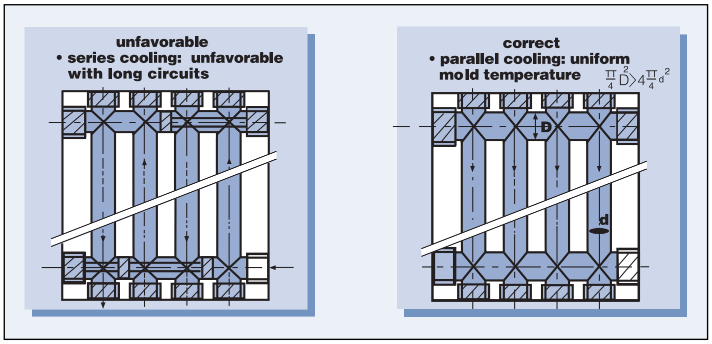


Figure 83 Example of core cooling by means of baffles: less effective heat discharge at core top

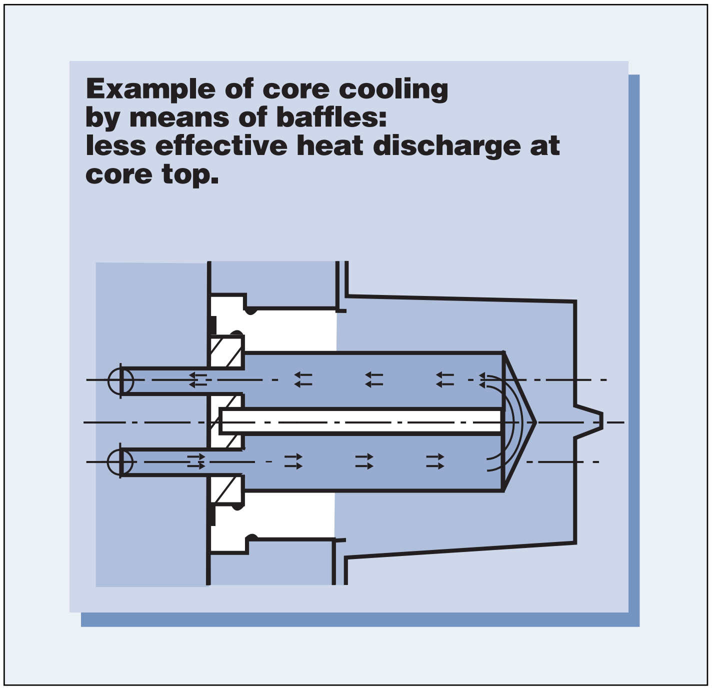
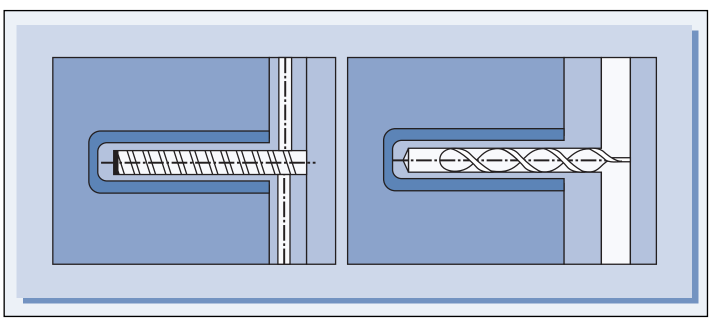


Figure 84 Single or double flight spiral cores.



Alternative cooling devices. For areas in the mold where it is not possible to use normal drilled cooling channels, alternative methods must be employed to ensure these areas are uniformly cooled with the rest of the part. The methods employed usually include baffles, bubblers, or thermal pins.

Baffles

Baffles and bubblers are sections of cooling lines that divert the coolant flow into areas that would normally lack cooling, e.g. cores. A baffle is actually a cooling channel drilled perpendicular to a main cooling line, with a blade that separates one cooling passage into two semi-circular channels. The coolant flows in one side of the blade from the main cooling line, turns around the tip to the other side of the baffle, and then flows back to the main cooling line.

This method provides maximum cross sections for the coolant, but it is difficult to mount the divider exactly in the center. The cooling effect and with it the temperature distribution on one side of the core may differ from that on the other side and this is the main disadvantage. The use of a helix baffle will solve the problem by conveying the coolant to the tip and back in the form of a helix. It is useful for diameters of 12 to 50 mm, and makes for a very homogeneous temperature distribution. Another logical development of baffles are single or double-flight spiral cores (see Figure 84).

Bubblers
A bubbler is similar to a baffle except that the blade is replaced with a small tube. The coolant flows into the bottom of the tube and "bubbles" out of the top, like a fountain. The coolant then flows down around the outside of the tube to continue its flow through the cooling channels. For slender cores this is the most effective form of cooling. The inner and outer diameters must be adjusted so that the flow resistance in both cross sections is equal. The condition for this is: Inner Diameter / Outer Diameter = 0.7.

Thermal Pins
A thermal pin is an alternative to baffles and bubblers. It is a sealed cylinder filled with a fluid. The fluid vaporizes as it draws heat from the tool steel and condenses as it releases the heat to the coolant. The heat transfer efficiency of a thermal pin is almost ten times as greater than a copper tube. For good heat conduction, avoid an air gap between the thermal pin and the, or fill it with a highly conductive sealant.

Cooling of large cores
For large core diameters (40 mm and larger), a positive transport of coolant must be ensured. This can be done with inserts in which the coolant reaches the tip of the core through a central bore and is led through a spiral to its circumference, and between core and insert helically to the outlet. This design weakens the core significantly.

Cooling of slender core
Cooling of cylinder cores and other round parts should be done with a double helix, as shown below. The coolant flows to the core tip in one helix and returns in another helix. The wall thickness of the core should be at least 3 mm in this case.

Figure 85 Example of core cooling (tube with flange): good heat discharge at core top.

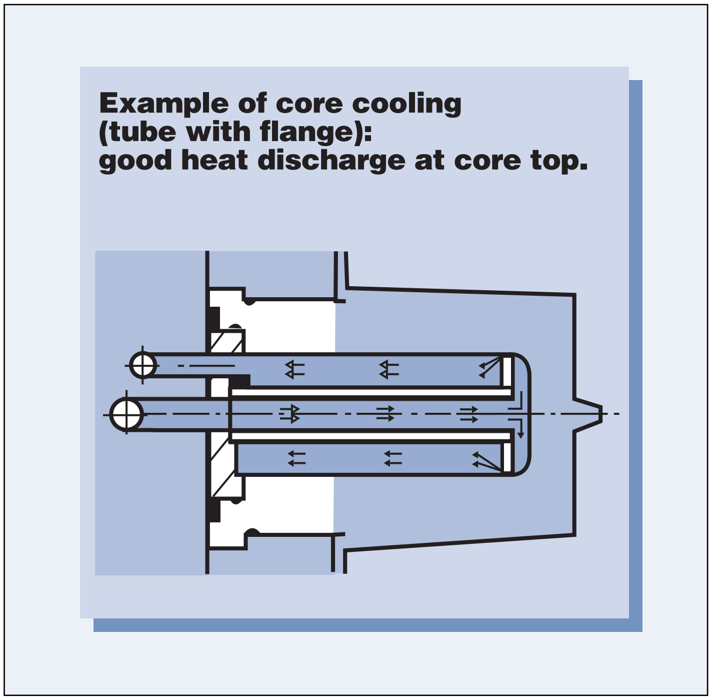


Figure 86 Cooling of large cores.

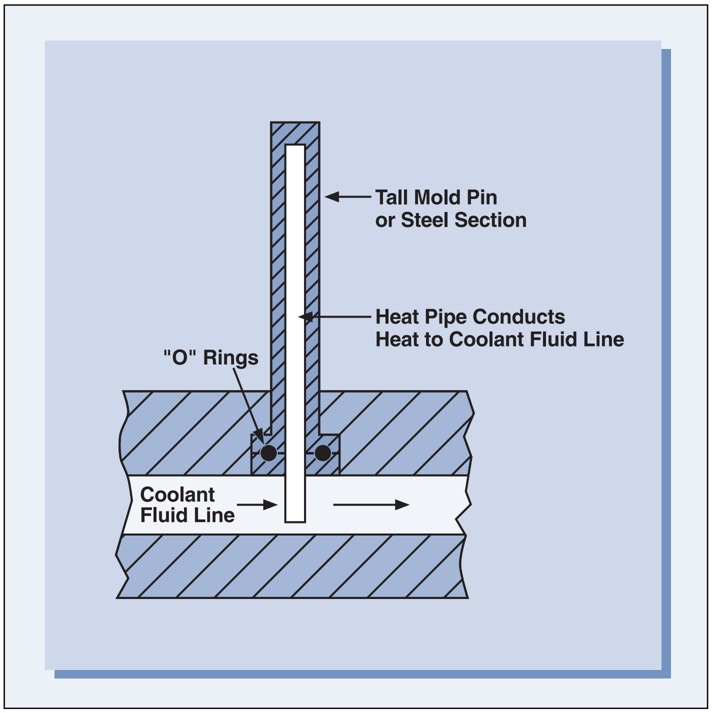


Figure 87 Cooling of slender core.

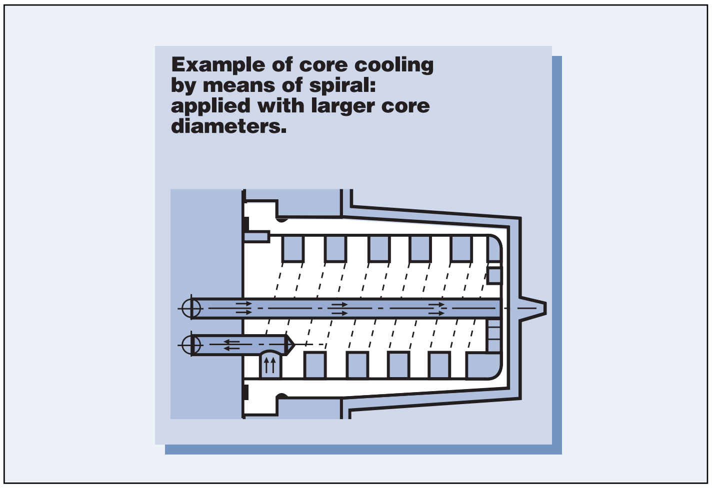


Figure 88 Cooling of slender core with inserts.

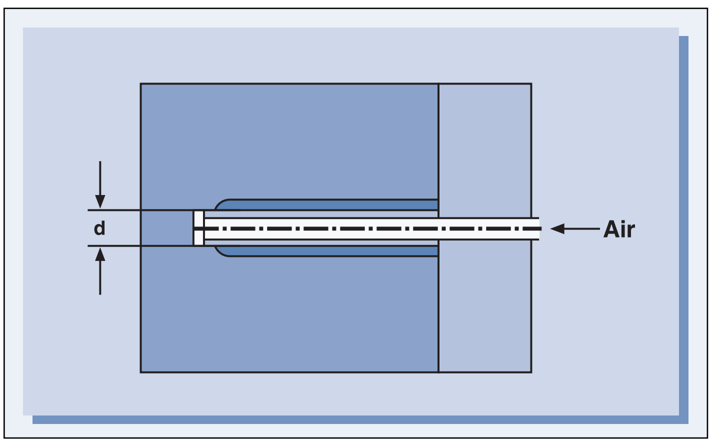


Figure 89 Cooling time equation.

$$\text{Cooling time} = \frac{(\text{Heaviest wall thickness})^2}{\text{Thermal diffusivity of polymer melt}}$$

$$\text{Cooling time} = \frac{(\text{Largest runner diameter})^{1.6}}{\text{Thermal diffusivity of polymer melt}}$$

where the thermal diffusivity of polymer melt is defined as

$$\text{Thermal diffusivity} = \frac{(\text{Thermal conductivity})^{1.6}}{(\text{Density})(\text{Specific heat})}$$

If the diameter or width is very small (less than 3 mm), only air-cooling is feasible. Air is blown at the cores from the outside during opening or flows through a central hole from inside. This procedure, of course, does not permit maintaining an exact temperature.

Better cooling of slender cores (those measuring less than 5 mm) is accomplished by using inserts made of materials with high thermal conductivity, such as copper or beryllium-copper materials. Such inserts are press-fitted into the core and extend with their base, which has a cross section as large as is feasible, into a cooling channel.

Cooling time. Theoretically, cooling time is proportional to the square of the heaviest part wall thickness or the power of 1.6 for the largest runner diameter (see Figure 89). In other words, doubling the wall thickness quadruples the cooling time.

Ejection System

The method of ejection has to be adapted to the shape of the molding to prevent damage. In general, mold release is hindered by shrinkage of the part on the mold cores. Large ejection areas uniformly distributed over the molding are advised to avoid deformations.

Several ejector systems can be used:

- Ejector pin or sleeve
- Blades
- Air valve
- Stripper plate

When no special ejection problems are expected, the standard ejector pin will perform well. In case of cylindrical parts like bosses a sleeve ejector is used to provide uniform ejection around the core pin. Blades are poor ejectors for a number of reasons: they often damage parts; they are prone to damage and require a lot of maintenance. Blade ejectors are most commonly used with ribbed parts.

A central valve ejector is frequently used in combination with air ejection on cup or bucket shaped parts where vacuum might exist. The air valve is thus only a secondary ejection device. A high-gloss surface can have an adverse effect on mold release because a vacuum may arise between cavity wall and the molding. Release can be improved by breaking the vacuum with an ejection mechanism.

A stripper plate or ring is used when ejector pins or valves would not operate effectively. The stripper plate is often operated by means of a draw bar or chain. Three-plate molds, as shown in Figure 91, have two parting lines that are used in multi-cavity molds or multiple gated parts. During the first opening stage automatic degating takes place when the parts are pulled away from the runners.

The opening stroke is limited by adjusting bolts, which also operate stripper plate A. Runners are stripped from slightly undercut cores at the injection side. Then, the mold is opened at the main parting line. Stripper plate B ejects the parts. The equation in Figure 92 can be used to calculate the require ejection force.

Figure 90 Blade ejectors.

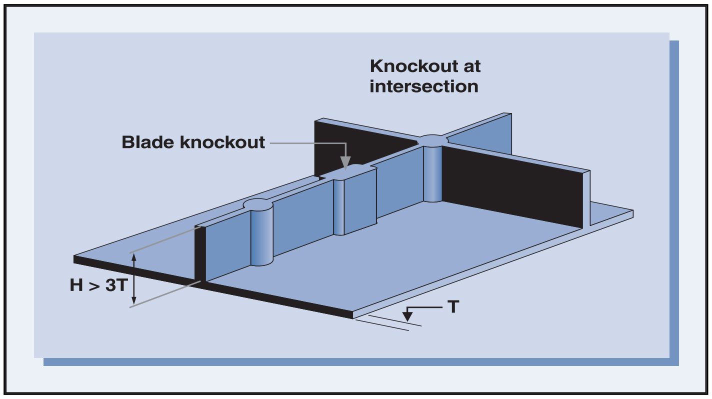


Figure 91 Three plate mold with two stripper plates for ejection.

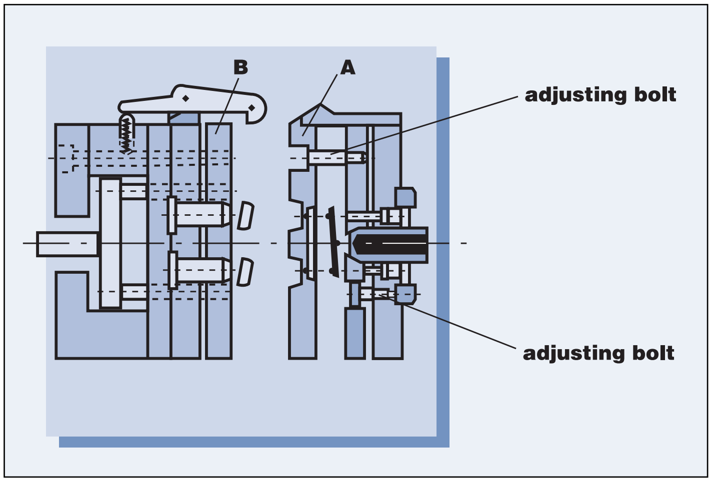
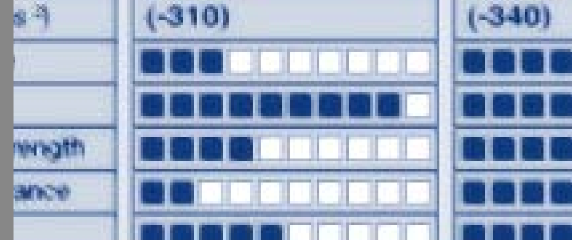


Figure 92 Ejection force equation.

$$P = \frac{S_t \times E \times A \times \mu}{d\,[d/2t - (d/4t \times V)]}$$

where:

- P = Ejection force (N)
- S_t = Thermal contraction of the plastic across diameter d = coefficient of thermal expansion x ΔT
- ΔT = Temperature difference (˚C)(transion temp - ejection temp)
- d = diameter of circle whose circumference is equal to the perimeter length of the molded part surrounding the male core (mm)
- E = Elastic modulus (MPa)
- A = Area of contact that shrinks onto core in the direction of ejection (mm^2)
- μ = Coefficient of friction, plastic/steel
- t = Thickness of molded part (mm)
- v = Poisson's ration of the plastic



Type of Tool Steel

For injection molds there are several steel types available. For long production runs a durable mold is required. The cost of tool steel is often not more than 10% of total mold cost. Important steel properties are:

- Ease of machining
- Dimensional stability after heat treatment
- Wear resistance
- Surface finish
- Corrosion resistance

Use of specific alloying elements like carbon may increase single properties, however, often at the cost of other properties. The table below shows some popular grades of mold tool steel.

Corrosion resistant hardened steels should be selected when conventional flame retardants are used. In the case of halogen free flame retardant DSM thermoplastics, standard steel types can be selected. Beryllium copper inserts may be used for improved cooling near hot spots. High heat conductivity is also required for gate drops in hot runner molds. Standardization of mold parts is growing, not only for ejector pins, leader pins and bushings, but also for mold plates and even complete mold bases. These standard mold bases require only machining of the cores, cavities and cooling channels and fitting of an ejection system. Advantages are:

- Cost savings (30-50%)
- Short delivery times
- Interchangeability
- Easy and rapid repair

Figure 93 Steel types for injection molds.

	A prehardened Ni-Cr-Mo steel, supplied at 290-330 Brinell, with excellent polishing and photo-etching properties. Suitable for a wide range of injection moulds, extrusion dies.	A prehardened stainless holder steel with excellent machinability, high tensile strength and good corrosion resistance.	A through-hardening stainless mold steel with good corrosion resisteance and very good polishability.	A versatile through-hardening 5% CR-mold and die steel with good wear and resistance and polishabilty.	A wear resistant through-hardening steel recommended for very long production runs of smaller, complicated moldings.
Properties	**45NiCr6**	**X36CrM01**	**X40Cr13**	**X40CrMO5**	**1X155CrVMO12**
Normal hardness 2)	(-310)	(-340)	52	52	60
Wear resistance	■■■□□□□□□□	■■■■□□□□□□	■■■■■■■□□□	■■■■■■■□□□	■■■■■■■■■■
Toughness	■■■■■■■■■□	■■■■■■■■□□	■■■■■□□□□□	■■■■■■□□□□	■■□□□□□□□□
Compressive strength	■■■■□□□□□□	■■■■■□□□□□	■■■■■■■□□□	■■■■■■■□□□	■■■■■■■■■□
Corrosion resistance	■■□□□□□□□□	■■■■■■■■□□	■■■■■■■■■□	■■■□□□□□□□	■■□□□□□□□□
Machining 3)	■■■■■□□□□□	■■■■■■□□□□	■■■■■■■■□□	■■■■■■■■■□	■■■■□□□□□□
Polishabilty	■■■■■■■■□□	■■■■□□□□□□	■■■■■■■■■□	■■■■■■■■□□	■■■■■□□□□□
Weldability	■■■■□□□□□□	■■■■□□□□□□	■■■■□□□□□□	■■■■□□□□□□	■□□□□□□□□□
Nitriding	■■■■■■□□□□	□□□□□□□□□□	□□□□□□□□□□	■■■■■■■■■■	■■■■■■■■■□
Photoetchabilty	■■■■■■■■■□	■■■■□□□□□□	■■■■■■■■□□	■■■■■■■■□□	■■■□□□□□□□
Norms					
DIN	(1.2710)	(1.2316)	1.2083	1.2344	1.2379 (1.2601)
USA AISI	P20	420F	420	(H13)	D2
Delivery HB	-310	340	215	175	210
Analysis (%)					
C	0.37	0.36	0.38	0.40	1.55
Cr	2.0	17.0	13.6	5.2	12.0

1) The properties of the main mold and holder steel grades have been rated from 1-10, where 10 is the highest rating. Such comparisons must be considered as approximate but can be a useful guide to steel selection.
2) Special process required.
3) rockwell C (Brinell).
Source properties: Steels for molds, Uddeholm

Surface Finish

A high-gloss surface finish may be achieved with proper molding conditions and polished mold cavities. High-gloss polished cavities require careful handling and protection during processing. Mold maintenance needs more frequent attention. Great care should be exercised when removing high-gloss parts from the mold to avoid scratches. Figure 94 provides an indication of the price index for the commonly used surface finish classes according ISO 1302.

For low gloss, semi-matt or matt surface finishes, the tool cavity needs treatment to obtain fine to very fine textured structures. A matt surface is obtained by vapor blasting techniques. Basic steel roughness should be N3 or better (ra < 0.1 mm).

Textured part surfaces have a special visual and haptic appearance, e.g. soft touch. Compared to other surface treatments, textures are relatively cheap. Their popularity is based on:

- Appearance (wood grain or leather)
- Functionality, e.g. anti-slip
- Masking of molding defects

Main texturing techniques include:

- Photochemical etching
- EDM
- Engraving
- Brushing
- Laser engraving

When high quality of textures are expected use a low alloy tool steel with a limited carbon content (< 0.45%). If nitriding is necessary, texturing should precede it. After long periods of use the mold surface deteriorates due to wear. Use of glass fibers will increase abrasion. Frequent checks of the surface condition are recommended.

Figure 94 Price index for various surface-finishing classes.

surface finish requirements

Roughness (µm)		Description	Price Index
≤0.05	NO-N2	high gloss, no visible scratches or flow lines	1000
0.1	N3	glossy, small, visable scratches acceptable	500
0.2	N4	"technical" finish	200
0.8	N5	no aesthetical requirements	100

surface finish requirements

Also differences in mold deposit tendency of materials can cause changes to the (local) mattness of parts, making cleaning at regular times necessary with some materials e.g. old PC/ABS formulations. Semi-crystalline thermoplastics are often less scratch resistant when very fine textures are used. Because of their good flow properties, the mold reproduction is better than that of amorphous thermoplastics. Micro-scopic ridges at the part surface may be easily damaged with a finger nail.

For untextured surfaces generally a minimum of 0.5 deg draft per side is recommended although there are exceptions when less may be acceptable. Polishing in draw line or using special surface treatments can achieve this. For textured sidewalls use an additional 0.4 deg draft per 0.1mm depth of texture is recommended.

Figure 95 Construction of a venting channel.

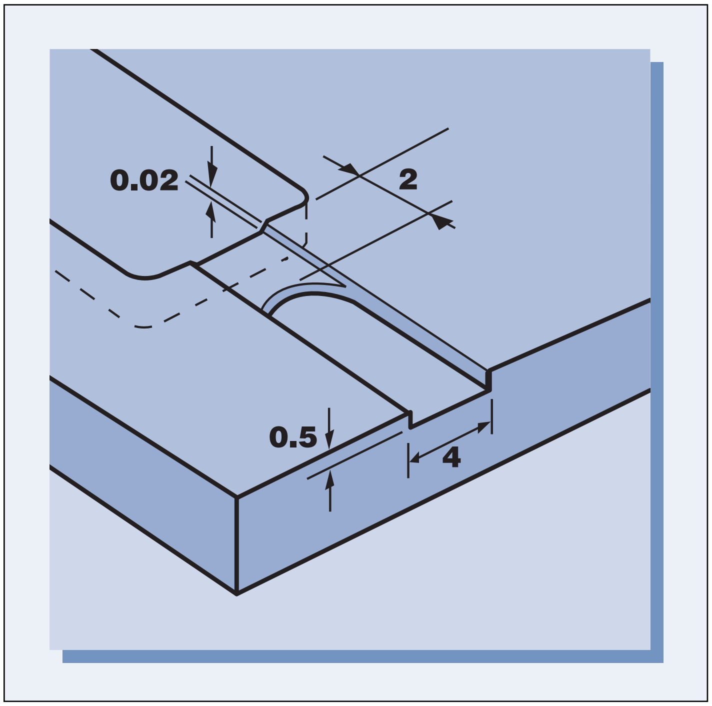
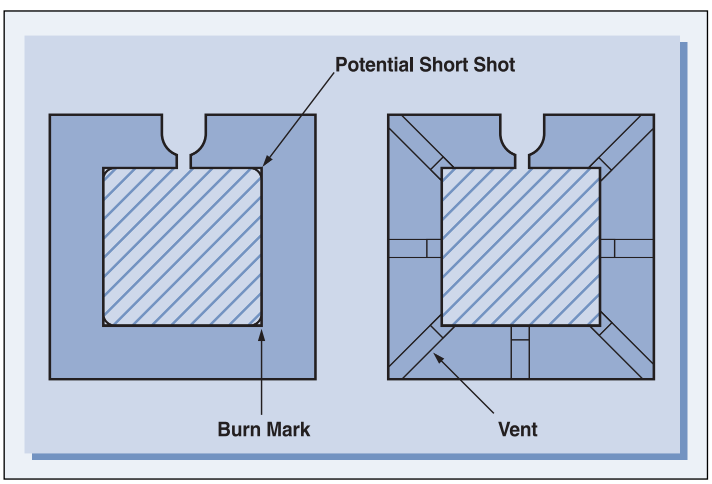


Figure 96 Venting locations.



Venting

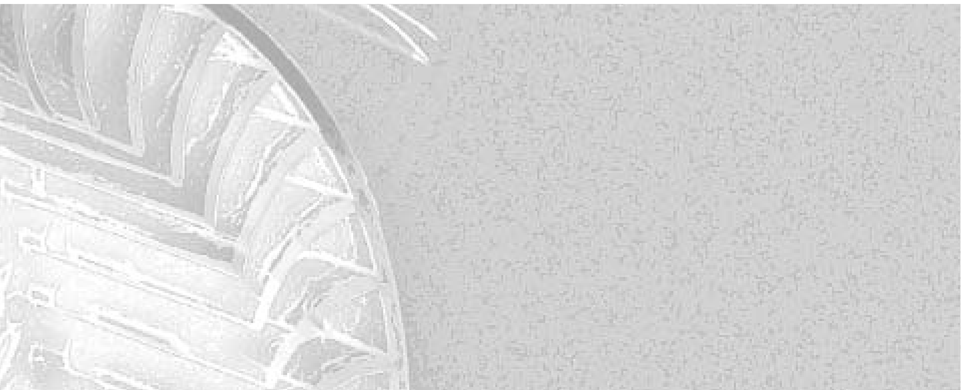
The displaced air in the mold cavity must be able to escape during mold filling process. If there are insufficient vents compression of air may take place. The pressure and local temperature rise quickly, potentially causing incomplete filling or even burning of the thermoplastic.

The number of vents is often limited by mold construction economics but ideally should be taken into account during the design stage. In general the higher the viscosity of a material the larger the vent dimensions. As with gates vents should be cut "steel safe" i.e. start at the minimum dimension and open the vent up gradually until the optimum molding is achieved. Too small and the vent will clog up and reduce or eliminate the ability to expel air from the mold cavity; too large and flash may be seen on the moldings. Dimensions of venting channels can be seen in Figure 95. The dimensions are chosen in such a way that air can escape without flash.

Vents can be placed anywhere along the parting line in particular they should be located in areas that are the last to fill in particular section of the mold. A reasonable spacing is every 25 mm (.98 in). If the air is trapped with no way out to the mold parting line, it is advisable to place a venting pin/ejector pin to permit the air escape through the clearance between pin and hole. Another option is to use sintered metal inserts, these inserts allow gas to pass into them without clogging up with the polymer. These inserts should only be used as a last resort and only on non-visual surfaces.

To summarize, inadequate venting may result in various molding failures:

- burnt spot
- weak and visible weld lines
- poor surface finish
- poor mechanical properties
- incomplete filling, especially in thin sections
- irregular dimensions
- local corrosion of the mold cavity surface



Unlimited. DSM

North America Headquarters
DSM Engineering Plastics
P.O. Box 3333
2267 West Mill Road
Evansville, IN 47732-3333
Tel: 1 800 333 4237
Fax: 1 812 435 7702
www.dsmep.com

Europe Headquarters
DSM Engineering Plastics
Poststraat 1
6130 AA Sittard
The Netherlands
Tel: 00 800 7466 3376
Fax: 00 800 3298 6376
www.dsmep.com

Asia Pacific Headquarters
DSM Engineering Plastics
11 Floor The Headquarters Building
No. 168 Middle Xi Zang Road
Shanghai 200001
China
Tel: 86 21 6141 8188
Fax: 86 21 6141 7010
www.dsmep.com

DSM Engineering Plastics

1-800-333-4237 - www.dsmep.com

Printed in the USA 02/05 1,000