3 – Moulding Considerations

Uniform Walls

Uniform wall thickness in plastic part design is critical. Non-uniform wall thickness can cause serious warpage and dimensional control problems. If greater strength or stiffness is required, it is more economical to use ribs than increase wall thickness. In parts requiring good surface appearance, ribs should be avoided as sink marks on the opposite surface will surely appear. If ribbing is necessary on such a part, the sink mark is often hidden by some design detail on the surface of the part where the sink mark appears, such as an opposing rib, textured surface, etc.

Even when uniform wall thickness is intended, attention to detail must be exercised to avoid inadvertent heavy sections, which can not only cause sink marks, but also voids and non-uniform shrinkage. For example, a simple structural angle (Fig. 3.01) with a sharp outside corner and a properly filleted inside corner could present problems due to the increased wall thickness at the corner. To achieve uniform wall thickness use an external radius as shown in Fig. 3.02.

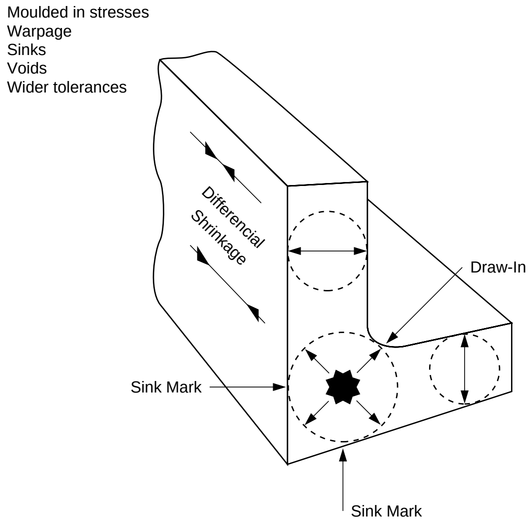


Fig. 3.01 **Effects of non-uniform wall thickness on moulded parts**

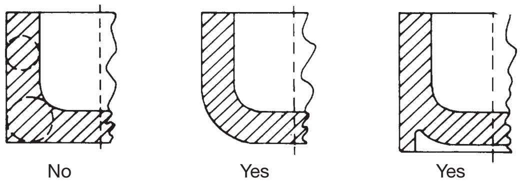


Fig. 3.02 **Outside corner design**

Configurations

Other methods for designing uniform wall thickness are shown in Fig. 3.03 and 3.04. Obviously there are many options available to the design engineer to avoid potential problems. Coring is another method used to attain uniform wall thickness. Fig. 3.04 shows how coring improves the design. Where different wall thicknesses cannot be avoided, the designer should effect a gradual transition from one thickness to another as abrupt changes tend to increase the stress locally. Further, if possible, the mould should be gated at the heavier section to insure proper packing (Fig. 3.05).

As a general rule, use the minimum wall thickness that will provide satisfactory end-use performance of the part. Thin wall sections solidify (cool) faster than thick sections. Fig. 3.06 shows the effect of wall thickness on production rate.

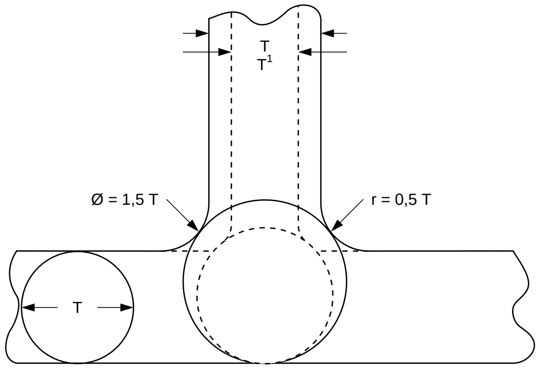


Fig. 3.03 **Rib dimensions**

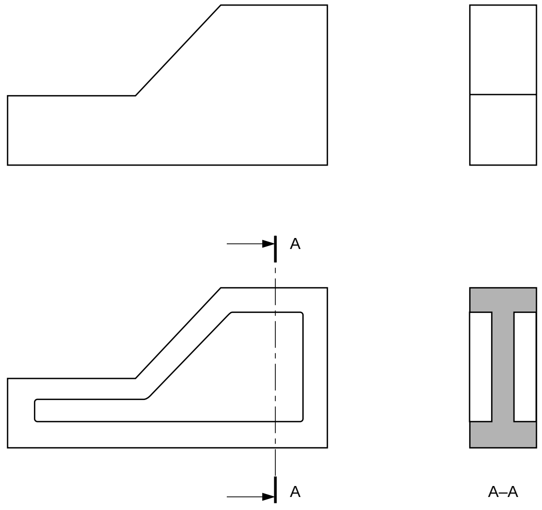


Fig. 3.04 **Design for uniform wall thickness**

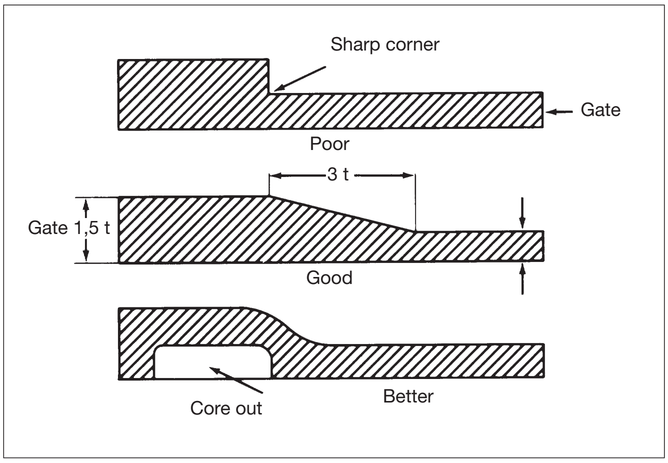


Fig. 3.05 **Wall Thickness Transition**

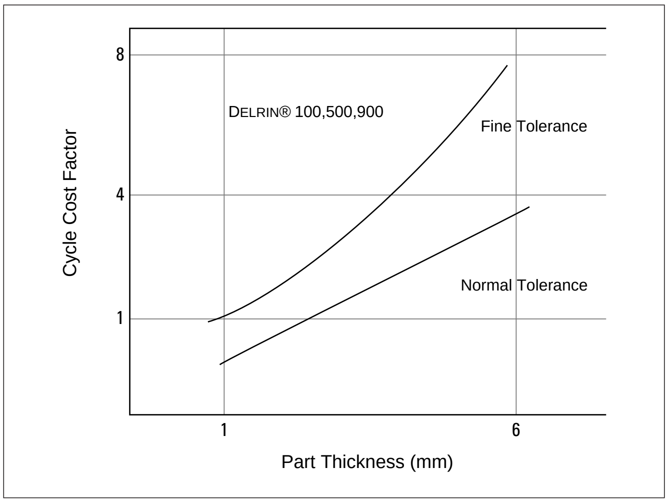


Fig. 3.06 **Cycle cost factor vs. part thickness**

Draft and Knock-Out Pins

Draft is essential to the ejection of the parts from the mould. Where minimum draft is desired, good draw polishing will aid ejection of the parts from the mould. Use the following table as a general guide.

Table 3.01 Draft Angle*

	Shallow Draw (Less Than 25 mm Deep)	Deep Draw (Greater Than 25 mm Deep)
CRASTIN® PBT	0 – ¼°	½°
DELRIN®	0 – ¼°	½°
ZYTEL®	0 – ⅛°	¼° – ½°
Reinforced Nylons	¼° – ½°	½° – 1°
Reinforced PBT	½°	½° – 1°
RYNITE® PET	½°	½° – 1°

* Smooth luster finish for textured surface add 1° draft per 0,025 mm depth of texture.

When knock-out pins are used in removing parts from the mould, pin placement is important to prevent part distortion during ejection. Also an adequate pin surface area is needed to prevent puncturing, distorting or marking the parts. In some cases stripper plates or rings are necessary to supplement or replace pins.

Fillets and Radii

Sharp internal corners and notches are perhaps the leading cause of failure of plastic parts. This is due to the abrupt rise in stress at sharp corners and is a function of the specific geometry of the part and the sharpness of the corner or notch. The majority of plastics are notch sensitive and the increased stress at the notch, called the ‘‘Notch Effect’’, results in crack initiation. To assure that a specific part design is within safe stress limits, stress concentration factors can be computed for all corner areas. Formulas for specific shapes can be found in reference books on stress analysis. An example showing the stress concentration factors involved at the corner of a cantilevered beam is shown in Fig. 3.07.

It is from this plot that the general rule for fillet size is obtained: i.e., fillet radius should equal one-half the wall thickness of the part. As can be seen in the plot, very little further reduction in stress concentration is obtained by using a larger radius.

From a moulding standpoint, smooth radii, rather than sharp corners, provide streamlined mould flow paths and result in easier ejection of parts. The radii also give added life to the mould by reducing cavitation in the metal. The minimum recommended radius for corners is 0,5 mm and is usually permissible even where a sharp edge is required (Fig. 3.08)

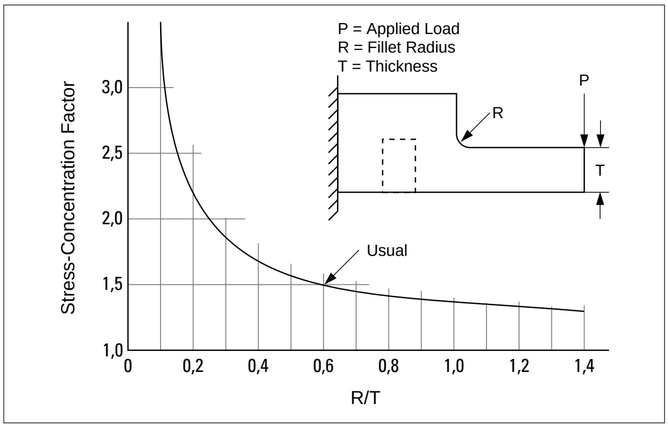


Fig. 3.07 **Stress concentration factors for a cantilevered structure**

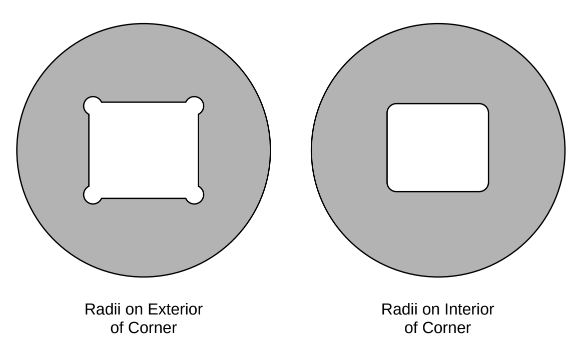


Fig. 3.08 **Use of exterior or interior Radii**

Bosses

Bosses are used for mounting purposes or to serve as reinforcement around holes. Good design is shown in Fig. 3.09.

As a rule, the outside diameter of a boss should be 2 to 3 times the hole diameter to ensure adequate strength. The same principles used in designing ribs pertain to designing bosses, that is, heavy sections should be avoided to prevent the formation of voids or sink marks and cycle time penalty.

Less good design of bosses can lead to sink marks (or even voids), see Fig. 3.10.

Weldlines in bosses should be avoided.

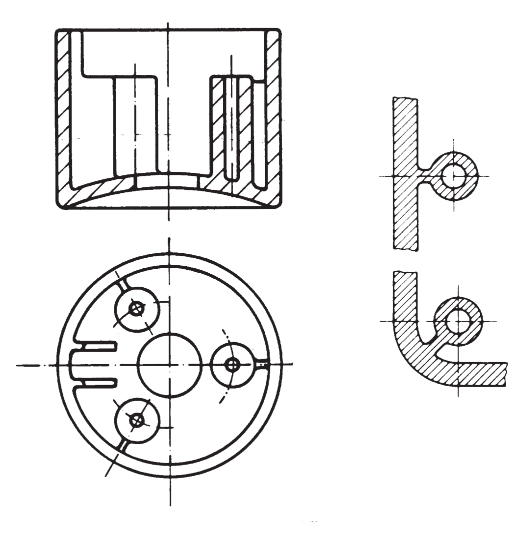


Fig. 3.09 **Good boss design**

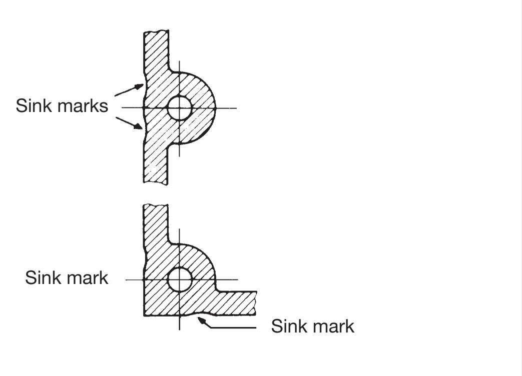


Fig. 3.10 **Less good boss design**

Ribbing

Reinforcing ribs are an effective way to improve the rigidity and strength of moulded parts. Proper use can save material and weight, shorten moulding cycles and eliminate heavy cross section areas which could cause moulding problems. Where sink marks opposite ribs are objectionable, they can be hidden by use of a textured surface or some other suitable interruption in the area of the sink.

Ribs should be used only when the designer believes the added structure is essential to the structural performance of the part. The word ‘‘essential’’ must be emphasized, as too often ribs are added as an extra factor of safety, only to find that they produce warpage and stress concentration. It is better to leave any questionable ribs off the drawing. They can easily be added if prototype tests so indicate.

For design with ribs, see chapter 4.

Holes and Coring

Holes are easily produced in moulded parts by core pins which protrude into the mould cavity. Through holes are easier to mould than blind holes, because the core pin can be supported at both ends. Blind holes formed by pins supported at only one end can be off-centre due to deflection of the pin by the flow of molten plastic into the cavity. Therefore, the depth of a blind hole is generally limited to twice the diameter of the core pin. To obtain greater hole depth, a stepped core pin may be used or a side wall may be counterbored to reduce the length of an unsupported core pin (Fig. 3.11).

Holes with an axis which runs perpendicular to the mould-opening direction require retractable core pins or split tools. In some designs this can be avoided by placing holes in walls perpendicular to the parting line, using steps or extreme taper in the wall (Fig. 3.12). Core pins should be polished and draft added to improve ejection.

Where weld lines caused by flow of melt around core pins is objectionable from strength or appearance standpoint, holes may be spotted or partially cored to facilitate subsequent drilling as shown in Fig. 3.13.

The guide below, referring to Figure 3.14, will aid in eliminating part cracking or tear out of the plastic parts.

d = diameter
b ≥ d
c ≥ d
D ≥ d
t = thickness

For a blind hole, thickness of the bottom should be no less than ⅙ the hole diameter in order to eliminate bulging (Fig. 3.15 A). Fig. 3.15 B shows a better design in which the wall thickness is uniform throughout and there are no sharp corners where stress concentrations could develop.

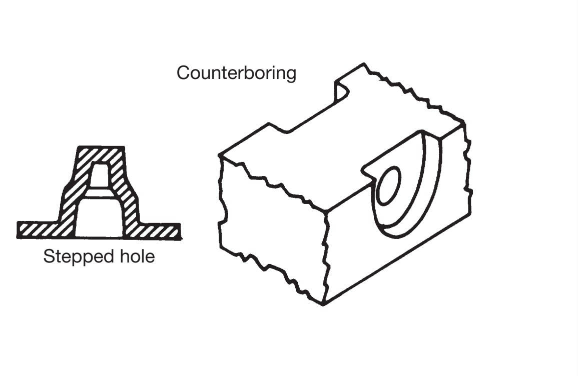


Fig. 3.11 **Blind hole with stepped core pin, counterboring**

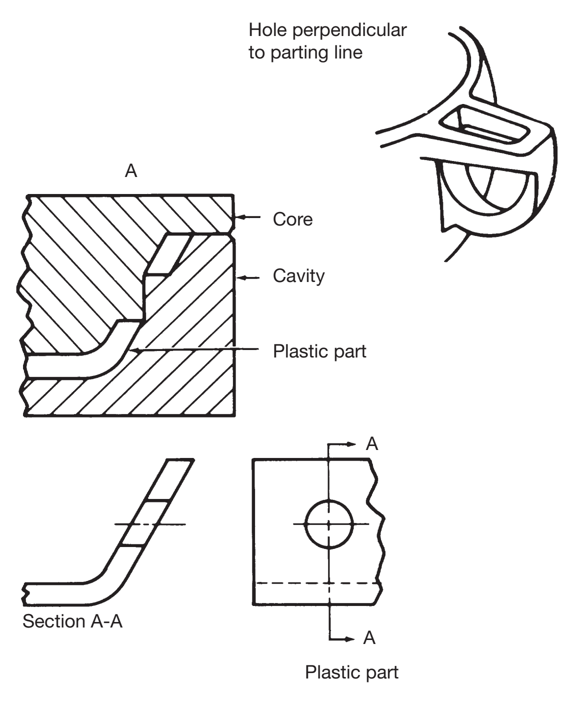


Fig. 3.12 **Avoiding side cores by special parting line design**

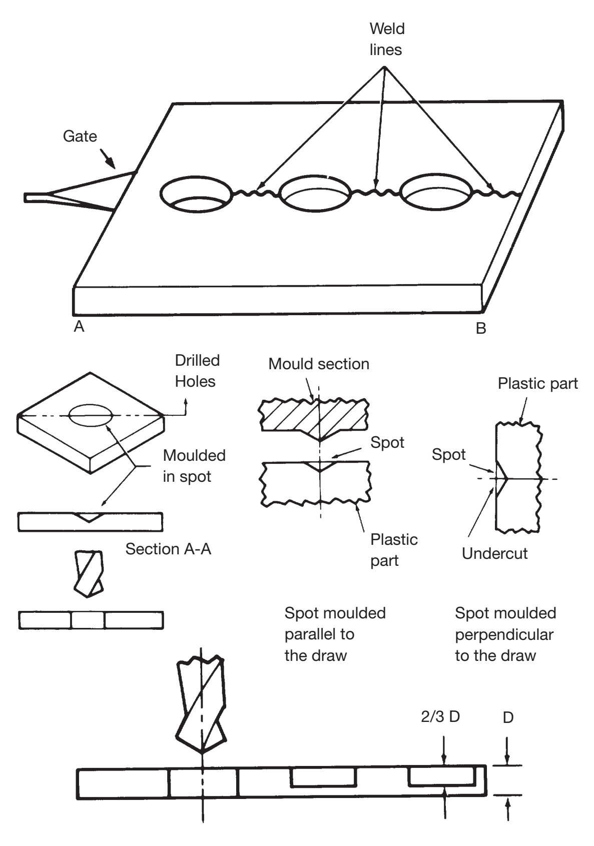


Fig. 3.13 **Drilled holes**

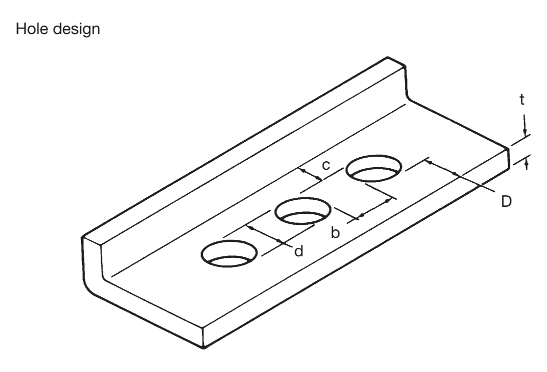


Fig. 3.14 **Hole design**

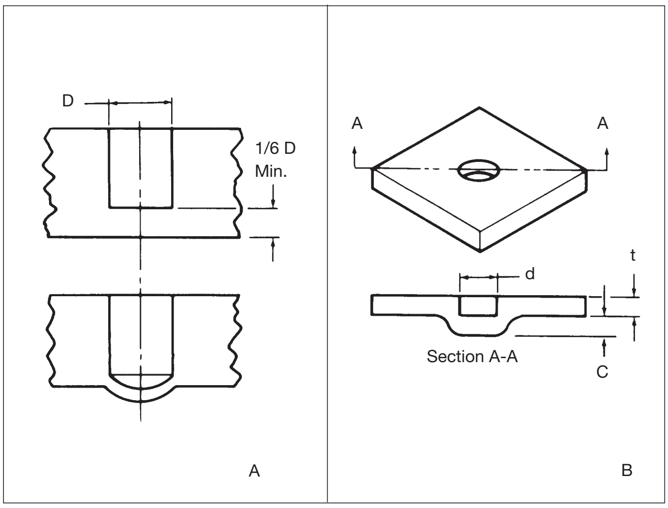


Fig. 3.15 **Blind holes**

Threads

When required, external and internal threads can be automatically moulded into the part, eliminating the need for mechanical thread-forming operations.

External Threads

Parts with external threads can be moulded in two ways. The least expensive way is to locate the parting line on the centreline of the thread, Fig. 3.16. It should be considered however that it is generally not possible to avoid an undercut in the parting line. This should lead to deformation of the thread on ejection. If this is not acceptable, or the axis of the thread is in the direction of mould-opening, the alternative is to equip the mould with an external, thread-unscrewing device.

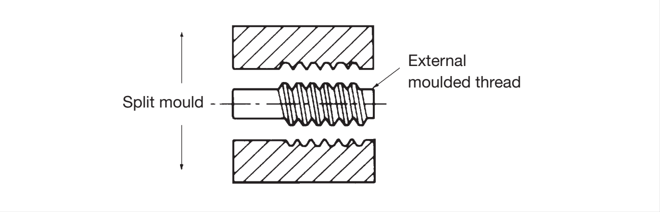


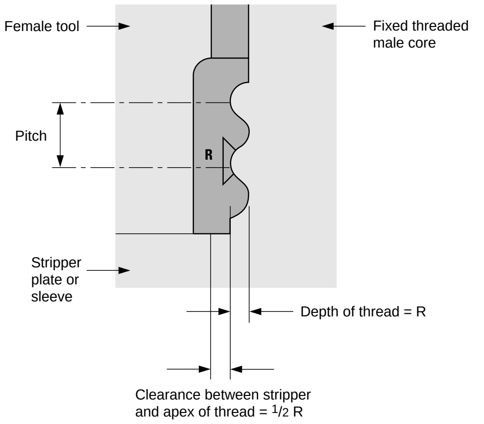
Fig. 3.16 **Moulding external threads without side core**

Internal Threads

Internal threads are moulded in parts by using automatic unscrewing devices or collapsible cores to produce partial threads. A third method is to use hand-loaded threaded inserts that are removed from the mould with the part.

Stripped Threads

When threaded parts are to be stripped from the mould, the thread must be of the roll or round type. The normal configuration is shown in Fig. 3.17 where R = 0,33 pitch. Requirements for thread stripping are similar to those for undercuts. Threaded parts with a ratio of diameter to wall thickness greater than 20 to 1 should be able to be stripped from a mould. Fig. 3.18 and 3.19 show the method of ejection from the mould.



Source: Injection-Mould Design Fundamentals,
A. B. Glanville and E. N. Denton, Machinery Publishing Co., London 1965

Fig. 3.17 **Stripping of roll-type thread**

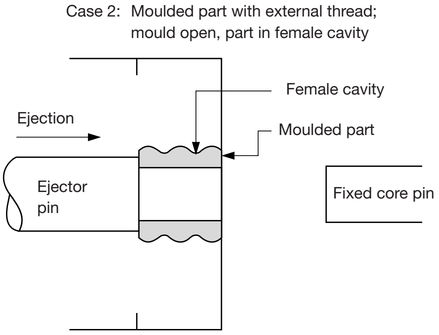


Fig. 3.18 **Mould-ejection of rounded thread-form undercuts – male**

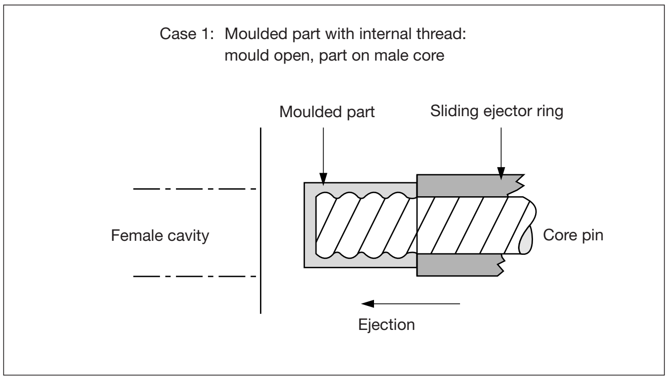


Fig. 3.19 **Mould-ejection of rounded thread-form undercuts – female**

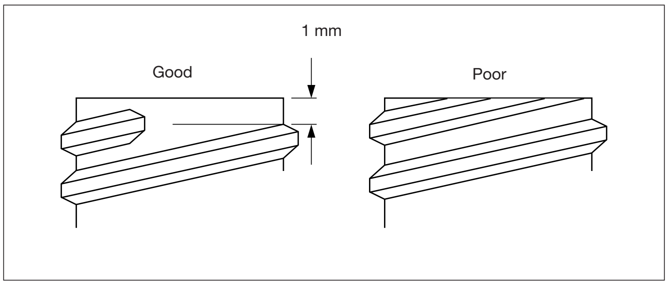


Fig. 3.20 **Correct termination of threads**

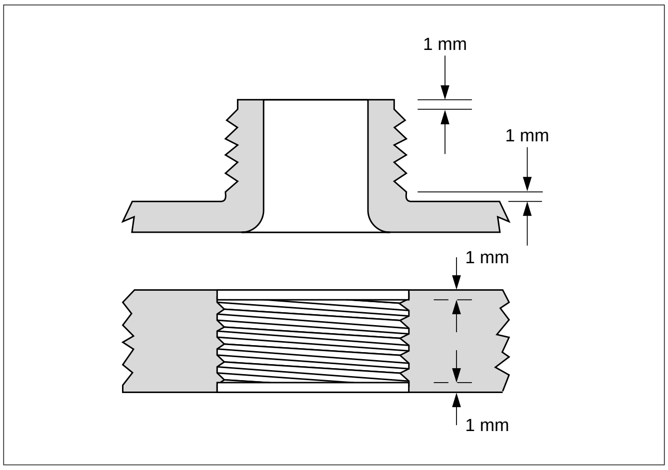


Fig. 3.21 **Suggested end clearance on threads**

Effect of Creep

When designing threaded assemblies of metal to plastic, it is preferable to have the metal part external to the plastic. In other words, the male thread should be on the plastic part. However, in a metal / plastic assembly, the large difference in the coefficient of linear thermal expansion between the metal and plastic must be carefully considered. Thermal stresses created because of this difference will result in creep or stress relaxation of the plastic part after an extended period of time if the assembly is subject to temperature fluctuations or if the end use temperature is elevated. If the plastic part must be external to the metal, a metal back-up sleeve may be needed as shown in Fig. 3.22.

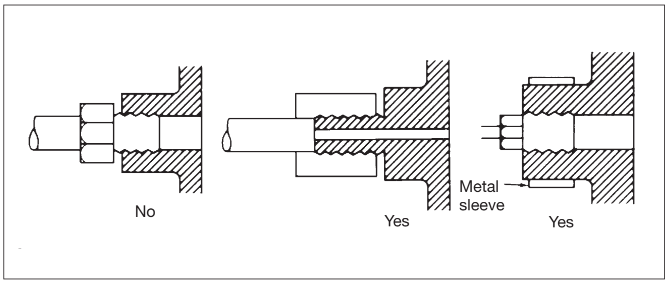


Fig. 3.22 **Metal-Plastic threaded joints**

Undercuts

Undercuts are formed by using split cavity moulds or collapsible cores.

Internal undercuts can be moulded by using two separate core pins, as shown in Fig. 3.23 A. This is a very practical method, but flash must be controlled where the two core pins meet.

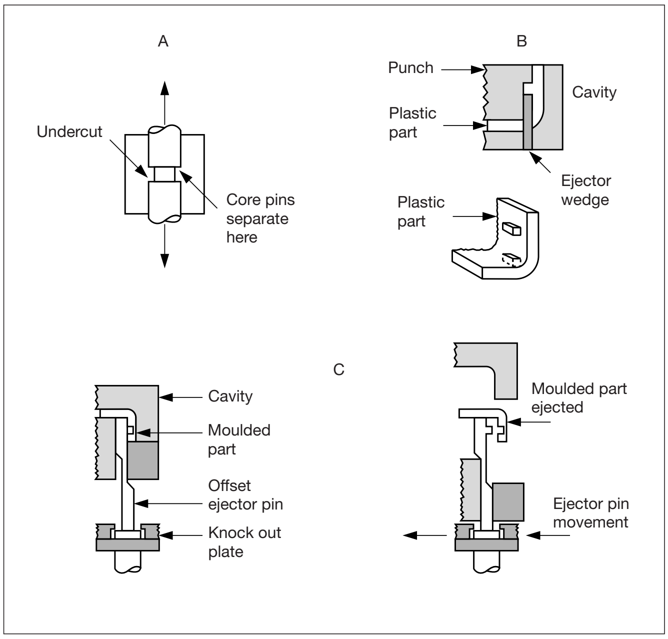


Fig. 3.23 **Undercut design solutions**

Fig. 3.23 B shows another method using access to the undercut through an adjoining wall.

Offset pins may be used for internal side wall undercuts or holes (Fig. 3.23 C).

The above methods eliminate the need for stripping and the concomitant limitation on the depth of the undercut.

Undercuts can also be formed by stripping the part from the mould. The mould must be designed to permit the necessary deflection of the part when it is stripped from the undercut.

Guidelines for stripped undercuts for specific resins are:

- *Delrin® Acetal Resin* – It is possible to strip the parts from the cavities if undercuts are less than 5% of the diameter and are beveled. Usually only a circular shape is suitable for undercut holes. Other shapes, like rectangles, have high stress concentrations in the corners which prevent successful stripping. A collapsible core or other methods described previously should be used to obtain a satisfactory part for undercuts greater than 5%.

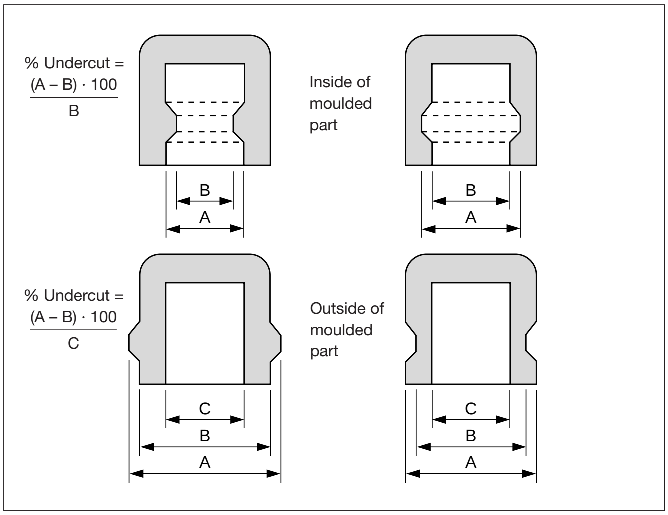


Fig 3.24 **Allowable undercuts for Zytel®**

- *Zytel® Nylon Resin* – Parts of Zytel® with a 6%-10% undercut usually can be stripped from a mould. To calculate the allowable undercut see Fig. 3.24. The allowable undercut will vary with thickness and diameter. The undercut should be beveled to ease the removal from the mould and to prevent over-stressing of the part.
- *Reinforced Resins* – While a collapsible core or split cavity undercut is recommended for glass-reinforced resins to minimize high stress conditions, carefully designed undercuts may be stripped. The undercut should be rounded and limited to 1% if stripping from a 40° C mould; or 2% from a 90° C mould.

Moulded-in Inserts

Inserts should be used when there is a functional need for them and when the additional cost is justified by improved product performance. There are four principal reasons
for using metal inserts:

- To provide threads that will be serviceable under continuous stress or to permit frequent part disassembly.
- To meet close tolerances on female threads.
- To afford a permanent means of attaching two highly loaded bearing parts, such as a gear to a shaft.
- To provide electrical conductance.

Once the need for inserts has been established, alternate means of installing them should be evaluated. Rather than insert moulding, press or snap-fitting or ultrasonic insertion should be considered. The final choice is usually influenced by the total production cost. However, possible disadvantages of using moulded-in inserts other than those mentioned previously should be considered:

- Inserts can ‘‘float’’, or become dislocated, causing damage to the mould.
- Inserts are often difficult to load, which can prolong the moulding cycle.
- Inserts may require preheating.
- Inserts in rejected parts are costly to salvage.

The most common complaint associated with insert moulding is delayed cracking of the surrounding plastic because of moulded-in hoop stress. The extent of the stress can be determined by checking a stress / strain diagramme for the specific material. To estimate hoop stress, assume that the strain in the material surrounding the insert is equivalent to the mould shrinkage.
Multiply the mould shrinkage by the flexural modulus of the material (shrinkage times modulus equals stress).
A quick comparison of the shrinkage rates for nylon and acetal homopolymer, however, puts things in better perspective. Nylon, which has a nominal mould shrinkage rate of 0,015 mm / mm* has a clear advantage over acetal homopolymer, with a nominal mould shrinkage rate of 0,020 mm / mm*. Cracking has not been a problem where moulded-in inserts are used in parts of Zytel® nylon resins.

The higher rate of shrinkage for acetal homopolymer yields a stress of approximate 52 MPa, which is about 75 per cent of the ultimate strength of the material.
The thickness of the boss material surrounding an insert must be adequate to withstand this stress. As thickness is increased, so is mould shrinkage. Due to stress relaxation stresses around inserts decrease with time.
After 100 000 hours, the 52 MPa stress will be reduced to approximately 15 MPa.

While this normally would not appear to be critical, long term data on creep (derived from data on plastic pipe) suggest the possibility that a constant stress of 18 MPa for 100 000 hours will lead to failure of the acetal homopolymer part. If the part is exposed to elevated temperatures, additional stress, stress risers or an adverse environment, it could easily fracture.

---

* 3,2 mm thickness – Recommended moulding conditions.

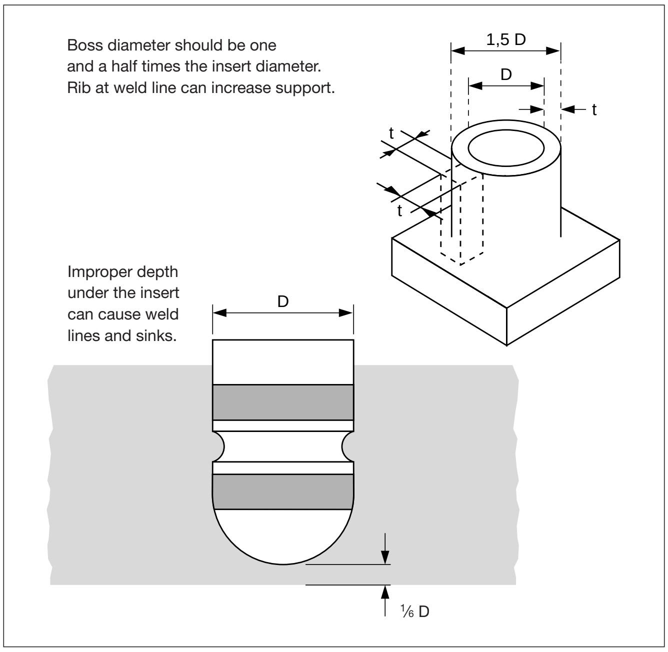


Fig 3.25 **Bosses and inserts**

Because of the possibility of such long-term failure, designers should consider the impact grades of acetal when such criteria as stiffness, low coefficient of friction and spring-like properties indicate that acetal would be the best material for the particular application.
These grades have a higher elongation, a lower mould shrinkage and better resistance to the stress concentration induced by the sharp edges of metal inserts.

Since glass and mineral reinforced resins offer lower mould shrinkage than their base resins, they have been used successfully in appropriate applications. Their lower elongation is offset by a typical mould shrinkage range of 0,3 to 1,0%.

Although the weld lines of heavily loaded glass or mineral-reinforced resins may have only 60 percent of the strength of an unreinforced material, the addition of a rib can substantially increase the strength of the boss (see Fig. 3.25).

Another aspect of insert moulding that the designer should consider is the use of nonmetallic materials for the insert. Woven-polyester-cloth filter material has been used as a moulded-in insert in a frame of glass-reinforced nylon.

Part Design for Insert Moulding

Designers need to be concerned about several special considerations when designing a part that will have moulded-in inserts:

- Inserts should have no sharp corners. They should be round and have rounded knurling. An undercut should be provided for pullout strength (see Fig. 3.25).
- The insert should protrude at least 0,4 mm into the mould cavity.
- The thickness of the material beneath it should be equal to at least one-sixth of the diameter of the insert to minimize sink marks.
- The toughened grades of the various resins should be evaluated. These grades offer higher elongation than standard grades and a greater resistance to cracking.
- Inserts should be preheated before moulding; 95° C for acetal, 120° C for nylon. This practice minimizes post-mould shrinkage, pre-expands the insert and improves the weld-line strength.
- A thorough end-use test programme should be conducted to detect problems in the prototype stage of product development. Testing should include temperature cycling over the range of temperatures to which the application may be exposed.

From a cost standpoint – particularly in high-volume, fully automated applications – insert costs are comparable to other post-moulding assembly operations. To achieve the optimum cost / performance results with insert moulding, it is essential that the designer be aware of possible problems. Specifying moulded inserts where they serve a necessary function, along with careful follow-up on tooling and quality control, will contribute to the success of applications where the combined properties of plastics and metals are required.

For the calculation of pull-out forces of metal inserts, see “Mechanical Fasteners”, chapter 9.

Tolerances

The tolerance which can be obtained by moulding is equal to:

$\Delta a = \pm (0{,}1 + 0{,}0015\ a)$ mm,

with a = dimension (mm)

In this formula, post moulding shrinkage, thermal expansion and/or creep are not considered and good moulding techniques are assumed to be used. For accurate moulding, 70% of the above tolerance can be obtained; for more coarse moulding, 140% should be taken.

For high accuracy moulding 40–50% of Δ a is applicable.

Shrinkage and Warpage

When plastic material is injected into a cavity, it starts to cool down, whereby its volume decreases. A measure for this volume decrease is given by the difference between melt density and solid density. As cooling rates in the cavity are very high and non-uniform, the frozen material will also incorporate internal stresses. These stresses may relieve after ejection from the cavity, which process can be accelerated by keeping the part at elevated temperatures.

Shrinkage can be defined by the following formula:

$S = (D - d) / D$ ($\times$ 100%).

D = dimension of mould cavity.

d = dimension of moulded part.

Shrinkage is usually not isotropic; it is direction dependant, in particular for glass-fibre reinforced materials. To be distinguished are:

- shrinkage in flow direction;
- shrinkage normal to flow;
- shrinkage in thickness direction.

The sum of these three shrinkages must be equal to the volumetric shrinkage of a material, which can be obtained from the difference between melt density and solid density, or from pVT-diagrams.

Besides on material, shrinkage is further dependant on processing conditions, (such as injection-speed, hold-pressure, hold-pressure time, runner/gate-dimensions and mould temperature), on part shape, (during injection the flow-direction may change) and part thickness, (thicker parts have usually a thicker central layer with less orientation).

The shrinkage contribution caused by stress relieve after ejection from the mould is called *Post Mould Shrinkage*.

Warpage is caused by internal stresses, which on their turn are the result of anisotropic shrinkage properties and non-uniform shrinkages.

Anisotropic shrinkage properties are mainly defined by the presence of reinforcements with high aspect ratios, (short glass fibres: ratio = 20), but also by different elastic behaviour of stretched cristals during filling, (residual stresses).

Non-uniform shrinkages can be the result of:

- anisotropic shrinkage;
- non-uniform thickness;
- non-uniform orientation;
- non-uniform mould temperatures;
- non-uniform hold-pressure (time).

Computer simulations have been developed to predict shrinkage and warpage. The reliability of the results of these predictions is increasing, particularly for parts made out of glass-fibre reinforced materials, as to day also methods are available to include shrinkage over the thickness, in which DuPont plays an important role.

Still one should be aware that it is very difficult to guarantee good results in all cases, as for instance the anisotropic shrinkage properties of a glass fibre reinforced material easily can be influenced by the screw and nozzle of an injection moulding machine, as well as by narrow gates. At these locations significant breakage of fibres may occur, thus affecting the anisotropic properties.