RUTLANDPLASTICS
Right People Right Partner

Injection Moulding is in our DNA
Design Guide

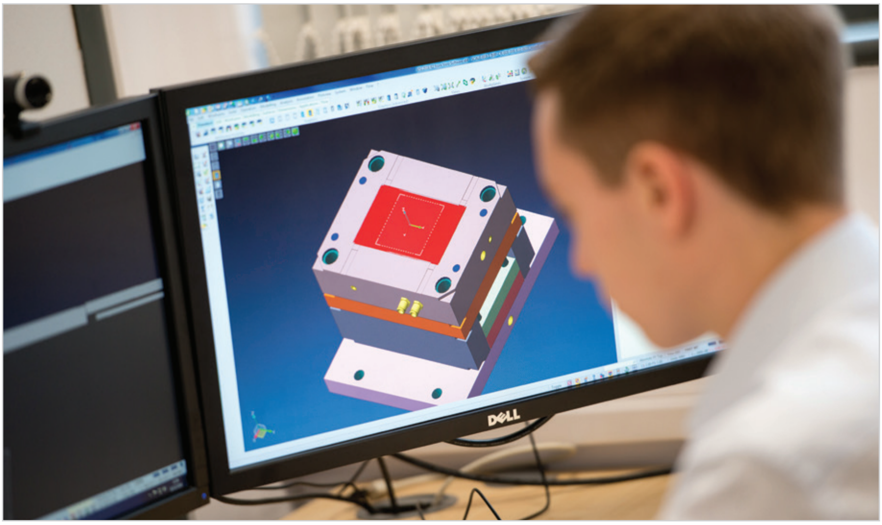
We at Rutland Plastics are frequently asked questions about basic design rules for plastic injection moulded parts or we receive drawings that require minor modifications to enable or improve mouldability. Whilst there are several books on the market to aid the designer, they run to several hundred pages and the majority of people do not have the time to read them. It is hoped that this concise guide will answer most of the frequently asked questions. This should not be seen as a replacement for seeking advice from Rutland Plastics but more of an assistance in getting the basics right in order to reduce the time from concept to production.

Advantages of plastic materials:

- Relatively easy to mould into complex shapes
- Lightweight with good strength: weight or stiffness: weight ratios. (Many consumers associate strength with weight which acts against plastics in some applications)
- Can be transparent
- Colours throughout the material so scratching or chipping of the product is less apparent as there is not a different colour below the surface
- Good chemical resistance for many plastics
- Varied mechanical performance
- Good thermal insulation

Contents

Design Development



The first step is to establish end-use requirements. This is probably the most important stage because if the specification is incomplete then the product or component may not be suitable for the application.

End use requirements fall into the major categories of functional, aesthetic and manufacturing related.

More specifically the following are factors to consider:

- Any special strength characteristics. When considering loading, for example, it is vital to be clear on the type of load, the rate at which the load may be applied, duration and frequency.
- Life expectancy
- Environmental. This can include exposure to chemicals/UV light, temperatures, relative humidity. Not forgetting any conditions that may apply during assembly and storage, such as exposure to paints, solvents, adhesives and even household cleaners.
- Aesthetic requirements including surface finish
- Assembly with other parts
- Dimensional requirements. It is necessary to specify critical dimensions and flatness, all with realistic tolerances.
- Cost considerations. Maximum cost for the product or component to be viable. Part of this calculation is likely production quantities.
- Legal requirements. Including food contact, flammability and any dimensional standards, e.g. for plumbing fittings.
- Need for recyclability

Appendix 1 is a form covering the most important design considerations.

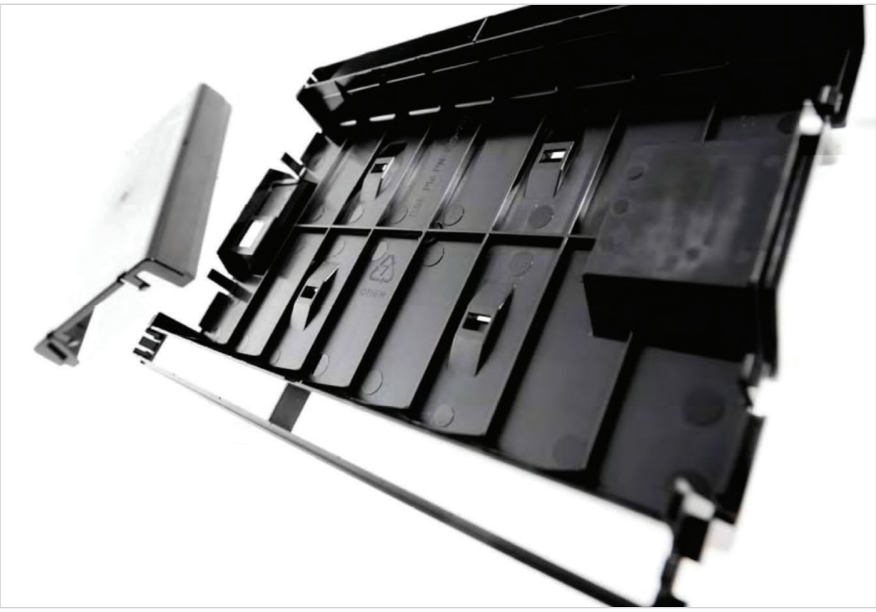
Design for Stiffness

When a part requires a certain degree of stiffness what needs to be considered are operating temperatures and length of time of the load. Also, is the load frequent or infrequent?

Material selection is important, possibly a filled material – talc, glass, etc. Stiffness is mainly increased in the direction of the glass fibres, so design with flow direction in mind.

But, design is also important. A key issue is whether the part is to be subjected to tensile or compression loading. For stiffness ribs should be added not additional section. Think of steel girders with 'I' and 'T' sections which are almost as rigid as solid beams but at a fraction of the weight and cost. It is also necessary to consider how many ribs are required to achieve the desired result. Ribs should not be too close together, especially if they are deep as this will make it difficult to cool the core that forms the rib pattern.

Side walls may be strengthened by the addition of buttress ribs. The same rules apply as for additional ribs as detailed below if sink marks on the outside of the part are to be avoided.



Stiffness can be increased by use of the following features:

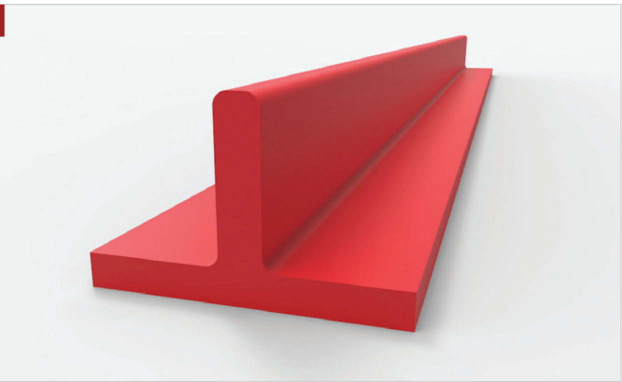
1. Ribs
Most commonly used. For a part that may bend ribs should be positioned perpendicular to the point of bending. For parts under tension, diagonal ribs are the most effective at increasing stiffness. In all instances deep ribs are more efficient than thick ribs.

2. V-grooves
Incorporated where significant increases in stiffness are required. However, often not used because they give uneven top and bottom surfaces. V-grooves should be perpendicular to the bend.

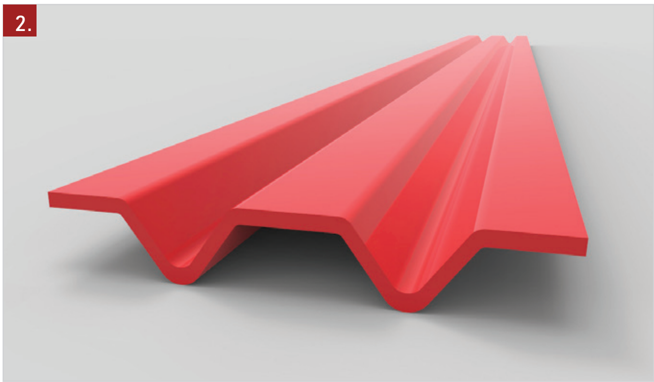
3. Corrugation
Similar to V-grooves.

4. Combinations
For a straight beam loaded in the middle corrugation is the stiffest followed by ribs then V- grooves assuming the same amount of material is used in each case. However, for a panel that could be loaded in different directions, say at right angles to each other, a rib form would then be the stiffest option.

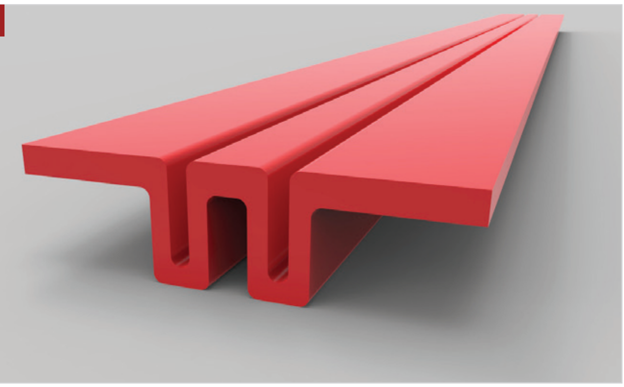
1.



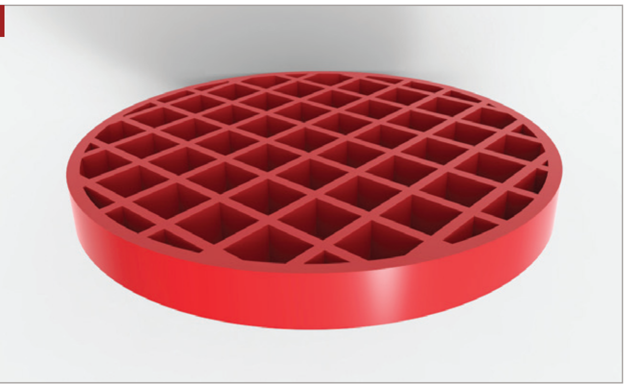
2.



3.



4.



Design for Strength

Designing for strength can be defined as:
'The maximum load that can be applied under certain conditions without causing failure to the part. Failure could be deformation, breakage or perhaps cracking, it would depend on the application. '

Factors Affecting Performance
As described under 'Designing for Stiffness', filled materials perform differently to unfilled and temperature has a bearing on performance also. Adding glass fibres will strengthen a material but also make it more brittle. Whereas an unfilled material is likely to bend at its limit a glass filled material is more likely to crack or snap.

With regards to temperature, extremely low temperatures will make a material more brittle whereas it will become softer and more ductile at higher temperatures. The relative strengths and tolerance of temperature extremes will vary from one material to another. When looking at replacing metal parts, for example, with a plastic moulded component, direct comparisons of materials are not possible. For instance, the tensile strength of a non-plastic material is similar across the normal temperature range. For most thermoplastics, the tensile strength will vary greatly with temperature changes.

Material performance can also be affected by chemicals, moisture and exposure to UV light.

Type of Strength Required

Part strength can mean different things subject to the conditions under which it is required to function. Ultimate strength could be based on the load a part is expected to carry – for example a plastic chair. This part would be tested well beyond what might be considered its maximum potential load to ensure that it would not fail. It may be that strength relates to deformation where no or very little deformation of a part is permitted. For example, a clip for retaining the cable of, say a vacuum cleaner. This needs to 'deform' enough to enable the cable to be removed and replaced but permanent deformation would render it useless.

For plastic parts the nature of stress is an important consideration. Plastic parts have a 'memory', that is to say that if the stress is applied for a relatively short period of time the part will return to its original shape once the stress is removed. However, if the part is stressed for a long time then 'creep' occurs – when the stress is removed the part will not return to its original shape.

Conclusions

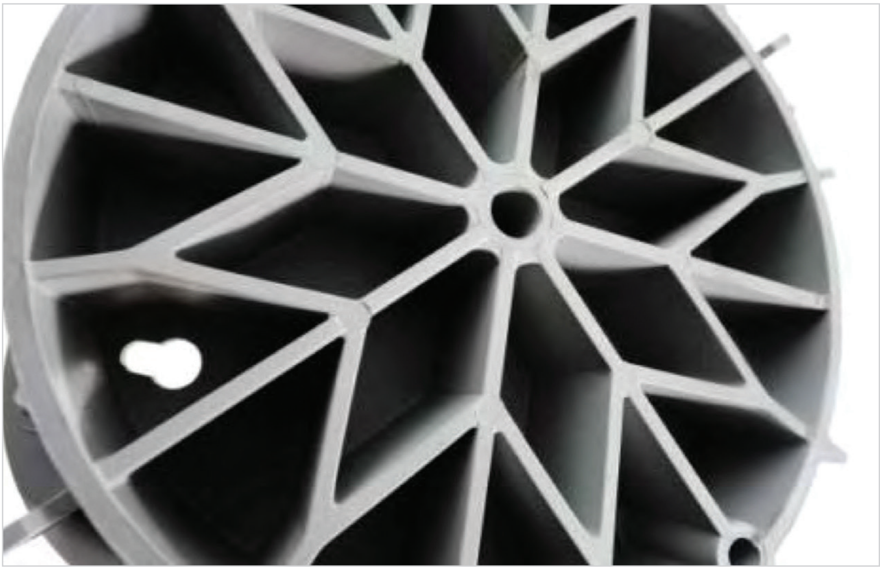
Once the particular type of strength requirement is defined it is then possible to identify a suitable material and design the part accordingly.

Design considerations for strength:

- Avoid sharp corners or notches, sharply angled wall intersections, large variances in sections and holes or inserts in the surface.
- Avoid situations where the part could be overloaded – over-tightening plastic threads for example.
- Ensure a good margin of safety as it is not possible to control all aspects of production an use. Also consider the worst case scenario for part failure.

Design for Impact Performance

A final consideration is the toughness of the part which could relate to the impact it has to withstand. For instance, car bumpers should flex and withstand low speed impact – typically less than 5mph, to avoid damage in a parking accident for example. This is an area where fillers can considerably increase the impact strength of a material



Various factors affect impact performance

- Type of material
- Wall thickness
- Geometric shape
- Temperature – the range it is required to work in and for how long at extremely high or low temperatures. Lower temperatures tend to make materials more brittle.
- Weight of loading
- Exposure to chemicals. Some plastics are vulnerable to attack by cleaning fluids, etc. It may be necessary for the product or component to be resistant to even more hostile chemicals.
- Flame retardancy
- Exposure to UV light

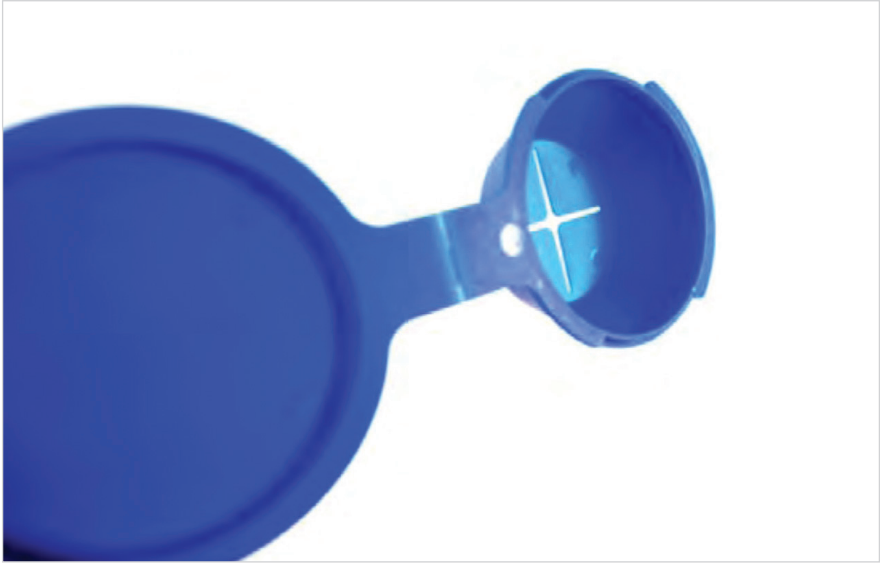
Design for Behaviour Over Time

When designing products consideration needs to be given to any stress that may be applied or other factors affecting performance over a period of time.

Creep

As mentioned earlier, plastic materials have a memory effect. If the plastic is stressed below its elastic limit for a short period of time, it will return to its original shape. If it is stressed for a long time, even below its elastic limit, the molecular arrangement within the part will change so that when the stress is removed the part will not return to its original shape. This phenomenon is known as creep and the amount of creep that occurs is dependent on the material stress applied, temperature and duration. Certain chemicals coming into contact with a polymer can also cause creep.

Creep can either be excessive deformation or an actual fracture of the component.



Fatigue

Fatigue is where stress applied is of a cyclical nature. The load is not generally great so that a single application would not result in failure of the part. An example of this would be a 'living hinge' – that is a thinner section of plastic joining two parts and acting as a hinge. Over time the stresses applied would be likely to result in the part failing. Design of the hinge and choice of polymer would be dependent upon the number of times the hinge is likely to be flexed.

Similar effects can be seen in parts subjected to vibration or repeated impacts.

Wear

Wear is another time related factor that would apply to components such as bearings or gears. It can be wear through friction at the point of contact between two materials, erosion where surfaces of different hardness are in contact or fatigue as a result of stress exceeding the endurance limit of the material.

Weathering

Within this section it is worth considering weathering as this affects behaviour over time. Many thermoplastics have poor long term resistance to weather. Ultraviolet radiation and oxidation can affect colour, transparency and material properties.

- Some plastics, such as acrylic, have excellent inherent properties suited to outside use. Other materials require the addition of stabilization or coatings to improve their performance in outdoor applications, e.g. polypropylene.

Design for Precision

Plastic mouldings shrink on cooling which can present problems. A thick section will either sink in so that the surface is not flat, as designed, and/or there will be voids within the material mass. There are no specific rules on wall section but the following guidelines are often stated:

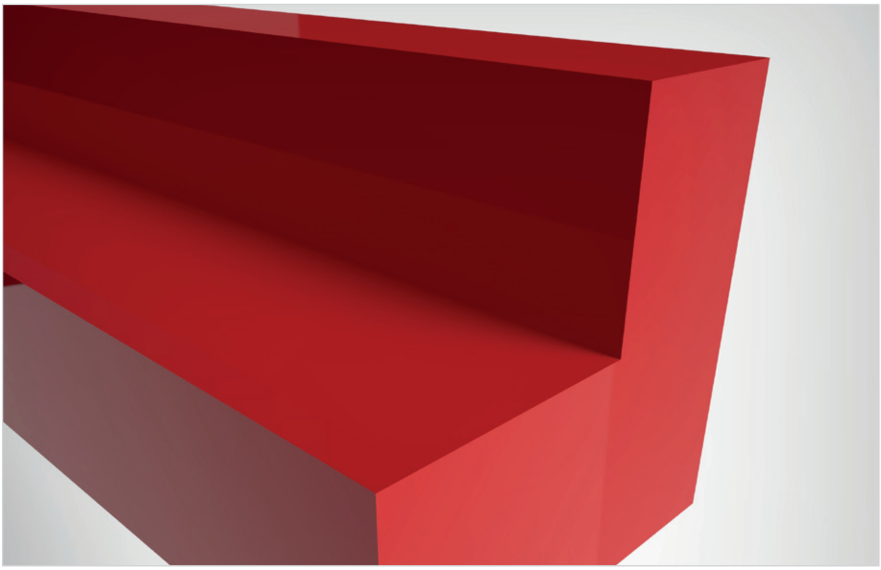
- 0.5mm – 5mm for unfilled materials
- 0.75mm – 3mm for reinforced materials

However, do not be misled. There are circumstances when much thicker sections are required and can be moulded. Rutland Plastics are specialists in thick section moulding so contact us for advice.

Warping

Frequently a part will shrink unevenly causing warping. Furthermore, parts may be heated during an assembly process, such as welding, or during curing following painting. This can cause both temporary and permanent dimension changes. Shrinkage also differs between in-flow and cross-flow.

Amorphous materials exhibit lower shrinkage than semi-crystalline. Amorphous are the hard plastics, such as ABS, polycarbonate, etc. Semi-crystalline are softer such as polyethylene and polypropylene. Fillers also affect shrinkage – tending to reduce it.



Wall thickness is a critical factor. Thin sections have a shorter cooling time so less opportunity for correction within the mould or by jigging. Varying thickness within a moulding also leads to distortion due to different shrinkages. If possible, a gradual transition between different sections should be used.

However, this differential shrinkage could also be used as a means of correcting warping by thickening sections at appropriate points.

Ribs can also be a problem, especially where they have been made thinner to prevent sink marks as once again there are variations in the thickness. Gas injection may be used to overcome the problem.

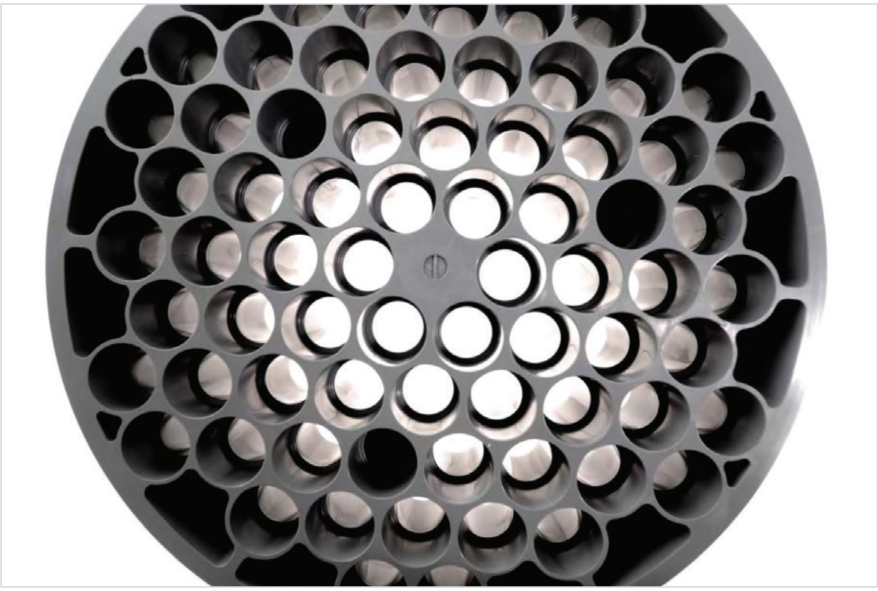
In addition to part design there are also a number of mould tool design and processing factors that need to be considered.

Tool design:

- Gate location
- Type and size of gate Cooling of the mould Ejection system

Processing:

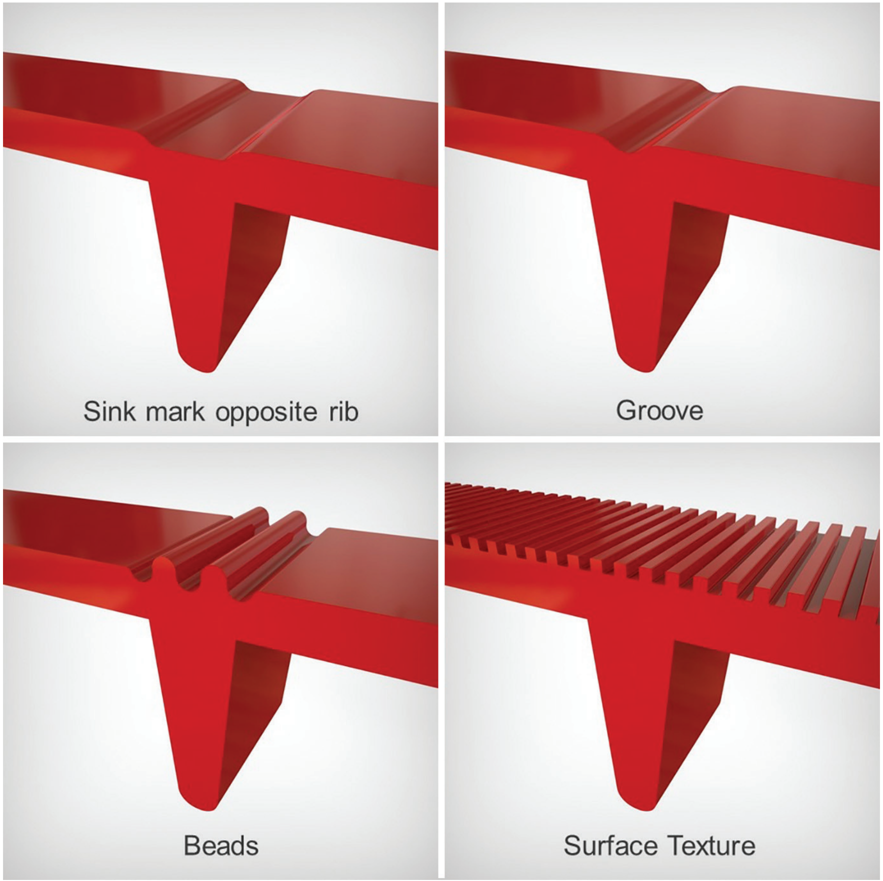
- Melt temperatures
- Mould temperatures (each half of the mould can be a different temperature to help overcome warping)
- Filling, packing and holding pressures and times
- Post moulding jigs



Design for Appearance

Although design contributes a great deal to the appearance of a part, material also has a bearing. Filled polymers tend to have a poor surface finish when compared to their unfilled equivalents. However, assessment of what is aesthetically pleasing is to some extent subjective.

Various surface defects can occur:
Sink marks typically occur over projections such as ribs or bosses. They are caused by localised thickening of the sections which results in an above average shrinkage. To minimise, attempts should be made to keep the thickness of ribs or bosses to 50% of that of the main wall. In certain circumstances gas assisted moulding may be used to minimise the effect such as in the case of TV surrounds. Sink marks may also be disguised by the use of a textured surface or a styling feature.



Each of the surface features have been exaggerated for clarity.

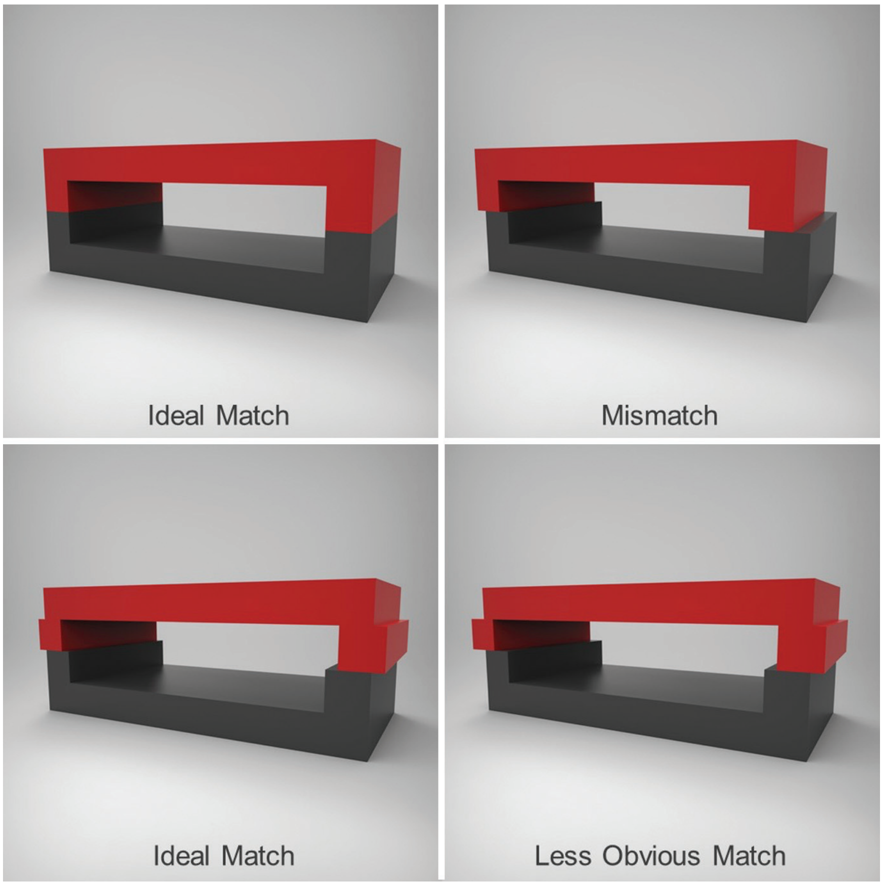
- **Weld Lines** occur where two flow fronts meet. Tool design can be critical in minimising the effect of these. Problems occur when there are holes in the component as the material has to flow around these and weld on the other side. In this instance there are not only small weld lines adjacent to the hole but there may also be flow lines where the smooth passage of the material has been interrupted. Weld lines can cause a physical weakening because the joint at the weld is usually weaker. They look like cracks on the surface of a moulded part.
- **Burning** occurs where there is insufficient venting at the edge of a moulding. As a result burn spots develop due to diesel effects. Air traps may also occur around ribs or sharp transitions – venting is then required around ejector pins or in some other way. Smoother transitions rather than sharp corners or steps can also alleviate the problem.
- **Voids** are air bubbles forming in the material due to thick sections. They are only visible in clear materials (unless they are particularly bad and break through to the surface) although they could well be present in any. This could lead to a weakening of the component. They are caused by excessive shrinkage. Wherever possible, excessively thick sections should be avoided. Also some materials are more prone to voids. Rutland Plastics is a specialist in thick section moulding and has several examples of void-free thick section mouldings (up to 110mm in one case).

There are other problems such as streaks, delamination, jetting and gate marks which are more the concern of tool designer rather than the product or component designer. Rutland Plastics can take your component design and advise on any necessary changes, tool design and manufacture is then managed in-house.

Assembly

If the part is not used in isolation then consideration must be made of assembly with other plastic or non-plastic parts. Methods of assembly include adhesive, welding, snap-fit, etc and can be manual or by automated process. It is possible to combine several features/parts into single moulding as well as adding threaded or other inserts. It is also necessary to consider other post-moulding operations such as painting or printing.

For assemblies there are various design features worth considering to cover instances where a mismatch is possible.



The aim of any project is to minimise the number of parts associated with a particular product. Fewer parts relates not only to lower costs but also easier assembly. Even eliminating a single screw can result in saving not just the cost of the screw but also the cost of assembly and handling. It could reduce the need for specialist equipment and scrap levels through damage caused by the incorrect fitting of the screw.

There are a number of assembly techniques:

- Press Fit
- Snap Fit
- Mechanical Fastening
- Hot Staking
- Welding
- Adhesive Bonding
- Solvent Bonding

Press Fit

Press Fit assemblies rely on interference between the parts to keep them joined together. Annular parts such as gears and pulleys, attached to shafts are a common example. Although simple there are potential problems. The degree of interference between the two parts is critical – too little and the joint is loose, too great and assembly becomes difficult and the material overstressed. Clearly, this type of assembly method is only viable where the associated manufacturing tolerances can be achieved and maintained. It should also be noted that where different materials are involved then changes in temperature could affect the fit.

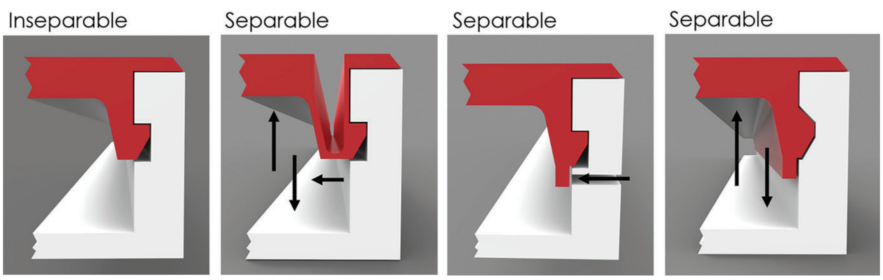
Snap fit

Snap fit is economical because it is a moulded feature and ideal for recyclability as there are no metal inserts or adhesives involved. All involve the same principle – a protruding feature on one part is deflected briefly on assembly to locate into a recess in the mating part. This method is not suited to parts requiring repeated assembly operations. Snap-fits can also be damaged due to incorrect handling especially if using a brittle or filled polymer.

There are three main types:

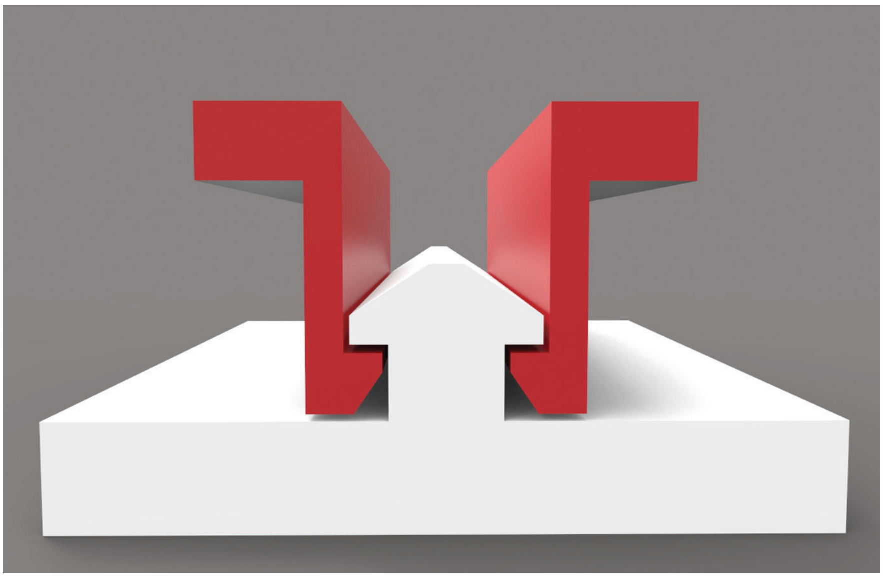
- Cantilever
- Cylindrical
- Spherical

Cantilever



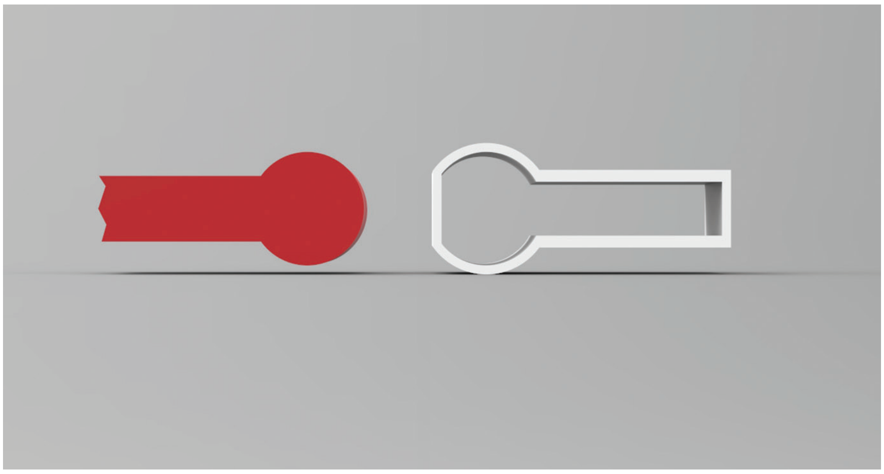
This works on the simple deflection of one part.

Cylindrical



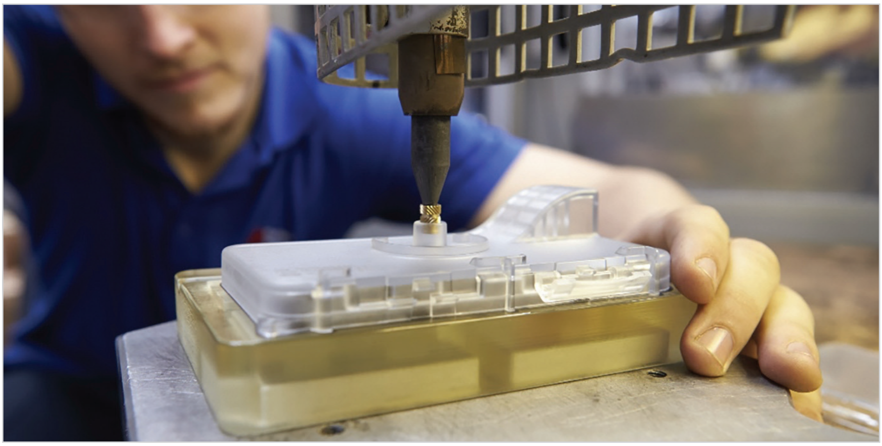
A much stiffer structure than cantilever, relying on the radial expansion of the female part. They are sometimes designed with a number of radial slots to make this expansion easier. This type of snap-fit is not suited to stiffer materials.

Spherical



An alternative version of the cylindrical snap-fit. The same principles apply.

Hot Staking



An alternative method of making a permanent joint is hot staking. A pillar on one moulding is inserted through a hole in the part it is to be joined to. This could be another moulding or a plate or printed circuit board, for example. Heat is then used to soften this pillar or stake and a tool used under pressure to form a head similar to a rivet. Frequently hot air is used hence the term hot air staking.

Advantages are similar to snap or press fits:

- They are an integral part of the moulding adding little or nothing to part or mould cost.
- They eliminate the need to buy and stock fasteners.
- Recycling is simplified as there is no need to remove foreign materials.

As thermoplastics are poor conductors of heat, when designing parts that are to be joined using this method in assembly it is better to have a greater number of small stakes rather than a few thick ones. Although the appearance is most commonly of a rivet head it is possible to achieve a flush finish by use of a countersunk head.

Mechanical Fasteners

There is a wide variety of different fasteners: self-tapping screws, press-in and mould-over inserts, clips, studs, rivets, etc they can either allow disassembly for repair or be permanent fixings. However, they can cause local stress in the moulding either through their insertion or the moulding of the holes to accommodate them. If the component is to be exposed to wide temperature variations it must be remembered that the vast majority of inserts are metal and so will have different expansion properties to the plastic material.

Welding

There are a number of different welding methods:

Ultrasonic welding

Is the most popular method and uses ultrasonic energy to create surface and intermolecular friction so generating heat and in turn forming a weld. Ultrasonic welding is suitable for most thermoplastics. Amorphous polymers are generally ideally suited, semi-crystalline polymers, on the other hand, are generally more difficult to weld ultrasonically.

Vibration welding

Involves rubbing together under pressure the two parts to be welded until sufficient heat is developed to cause the materials to melt at which point the motion is stopped and the parts allowed to cool forming the weld. As with ultrasonic welding, most thermoplastics can be vibration welded although amorphous materials tend to be easier to vibration weld than semi-crystalline polymers.

Spin welding

Is again a friction process and as the name suggests, is used to join parts with rotationally symmetrical joining surfaces. One part is rotated while the other is held firmly, pressure is applied and the motion stops once the material is in a molten state in the weld zone. This is the most efficient joining process for parts with circular cross sections. Cylindrical parts up to 600mm diameter with a joint area as large as 58cm2 have been successfully joined using this method.

Induction (Electromagnetic) welding

Uses inductive energy to achieve fusion temperature. Again, this method may be used with most thermoplastics. Before welding an electromagnetic welding material, such as a gasket, is placed between the two parts to be joined usually in a groove or other recess. During welding an electromagnetic field is formed which heats the electromagnetic sensitive material, the heat is transferred to the surfaces of the parts to be joined and the parts soften. Welding is completed with the application of pressure.

Resistance welding

Is similar to induction welding except that an electrically conductive wire or braid is used in the place of electromagnetic material. An electric current is applied to the wire or braid which then heats up and softens the thermoplastic material on the two areas to be joined which can then be welded with the application of pressure. It is a quick and simple method ideally suited to very large parts. The wire remains in place which may weaken the finished weld.

Hot tool welding

Uses electrically heated tooling to soften the plastic part surfaces to be joined. The tool is then removed and the parts pressed together before the surfaces cool. This method can be used for all sizes of parts and almost any thermoplastic material. It is particularly suited to semi-crystalline polymers such as polypropylene and polyethylene. Unfortunately, the cycle times for this type of welding can be long, especially for larger parts.

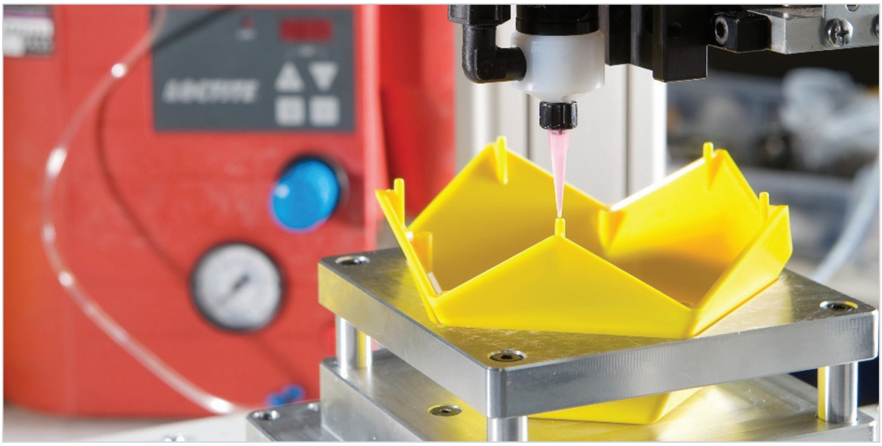
Hot gas welding

Is used for very large parts although not usually injection moulded components as there are better alternative methods.

Extrusion welding

Evolved from hot gas welding. A V-groove on the part to be joined is heated using hot air or gas to soften it. It is then filled under pressure with an identical material which has been plasticised via extrusion. Once deposited pressure is applied. Usually used for larger sheet type structures.

Adhesive Bonding



Advantages:

- Aesthetic and design flexibility
- Uniform stress distribution
- Can join dissimilar materials
- Can provide tight seal
- Flexible adhesives can compensate for thermal expansion mismatches between materials
- Flexible adhesives can dampen vibration
- Can be used with thin, flexible substrates
- Provide electrical and thermal insulation

As with the materials themselves, adhesives can be affected by environmental factors and time. Unlike other fastening methods it can take time for the adhesively bonded joint to reach full strength.

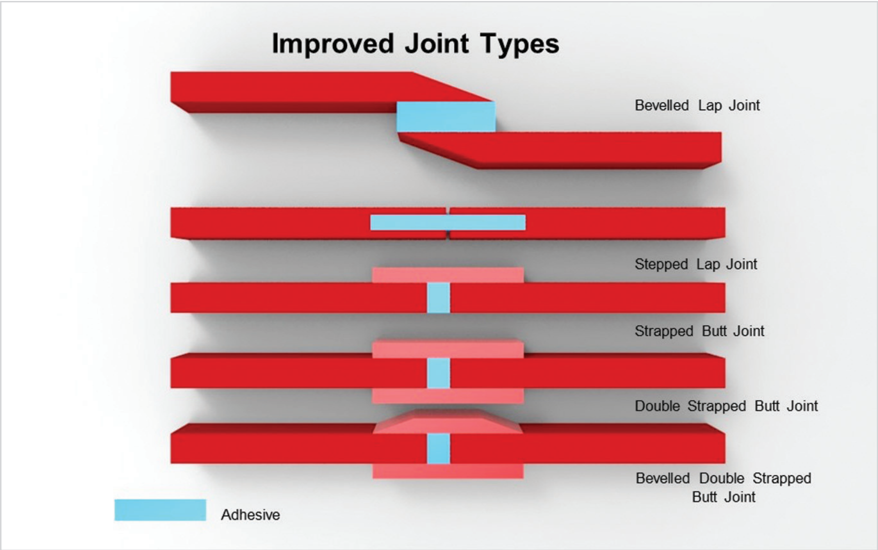
Solvent Bonding

This method can only be used with certain types of thermoplastics and is mostly used with amorphous materials. This is more of a welding process whereby the solvent softens the surfaces of the two parts to be joined; they are then clamped together while the solvent evaporates.

This is a simple relatively inexpensive method but it does require the parts to be joined not to be warped and be moulded to relatively tight tolerances.

Joint designs tend to be variations of two main types – lap and butt. A lap joint is where the two parts to be joined overlap whereas the butt involves the two parts to be joined end to end. Both types of joint have variations that give improved performance. Lap joints may be stepped or beveled and butt joints may have some form of strap applied over the joint. The use of a strap in this manner adds to cost but can make for a design feature.

Cost – avoid 'over-engineering'. It is necessary to consider likely volumes and life expectancy of the product. A more simple design could be cost effective whereas a more complex one may be uneconomic.



Design for Mouldability

The aim for the most cost effective solution is to design parts in their finished form, i.e. without the requirement for secondary operations and with no waste of materials.

The main advantages of injection moulding are:

- Production of highly complex parts is possible
- Tight tolerances are repeatable
- Features such as bosses, snap-fits, undercuts and holes can be moulded in High quality surface properties are possible
- Automation is easier on long production runs

There are various design considerations:

Nominal Wall Thickness

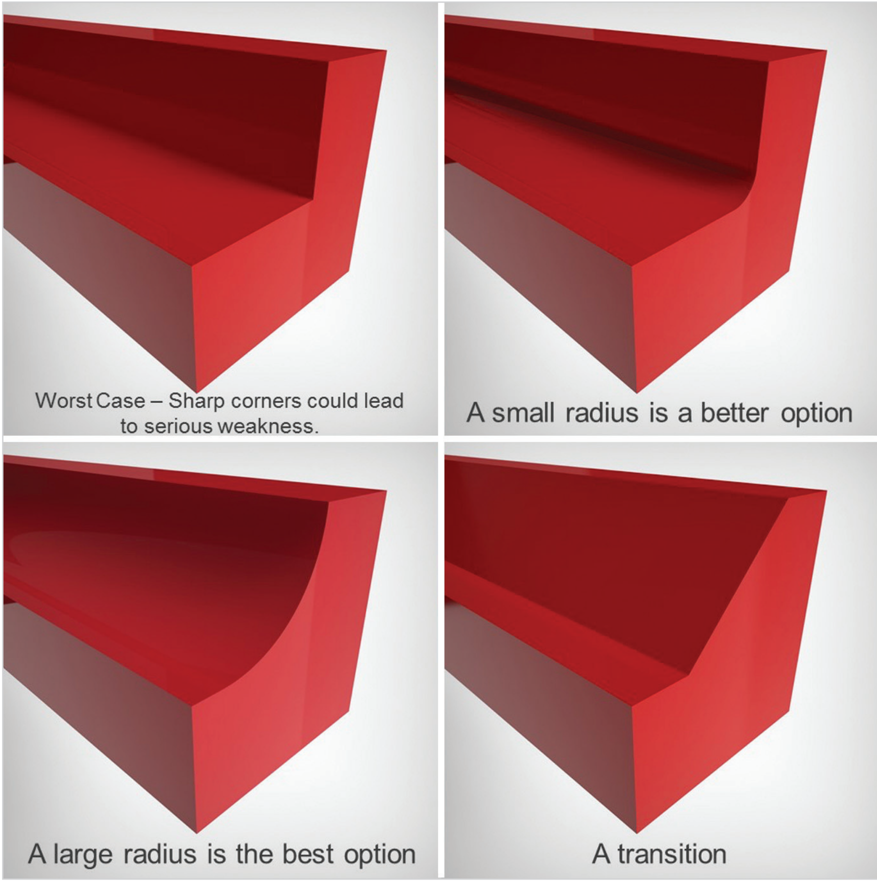
It is important that this should be correctly designed. A wall section that is too thin can lead to structural failure or poor insulation characteristics. A wall section that is too thick can result in appearance defects and an overweight or over-engineered part. With the latter point it is also worth remembering that wall thickness governs the moulding cycle time – the thicker the section the longer the cycle time and therefore the more expensive the part becomes. Furthermore, plastics shrink during cooling which in thick sections can result either in the surface of the part forming a sink mark or an internal void.

In most applications a thin, uniform wall with ribs is preferable to a thick wall. There are general guidelines for how thick a part should be, typically 0.75mm – 3mm for filled materials and 0.5mm – 5mm for unfilled. However, this does depend on the design and function of the part concerned.

It is possible to mould much thicker sections if required, Rutland Plastics has successfully moulded void free parts up to 120mm thick.

Gas assisted injection moulding can be used in some instances where ribbing is not possible or desired or where the general wall section is relatively thin and ribs cause surface imperfections, e.g. TV surrounds. Chemical foaming agents are also an option with thick section moulding. Where changes in thickness occur the aim should be to have the flow of the material from a thicker to a thinner section. Sharp corners should also be avoided as they cause stress which could lead to failure of the component. In cases where the thickness of a part must be increased, this should be less than 25%.

Radii



Radii are important to alleviate stress in the part. Generous radii should be used in areas of major change in section or direction. Furthermore, sharp corners should be avoided because:

- Facilitates ejection of the part from the mould. Sharp internal corners tend to stick in the mould as the part shrinks onto the core.
- Provides smooth flow. Resin flows more easily around smooth radiused corners than sharp ones.
- Reduces shear. Shear around sharp corners can degrade the material and cause aesthetic problems.

An internal radius should be at least half the wall thickness and preferably in the range of 0.6 to 0.75 times wall thickness. However, this can result in a thicker section at the corner which goes against what was stated in the previous section. The result can be longer cycle times and a greater risk of distortion and sink marks. In an ideal situation the internal radius should be matched externally so maintaining a constant section.

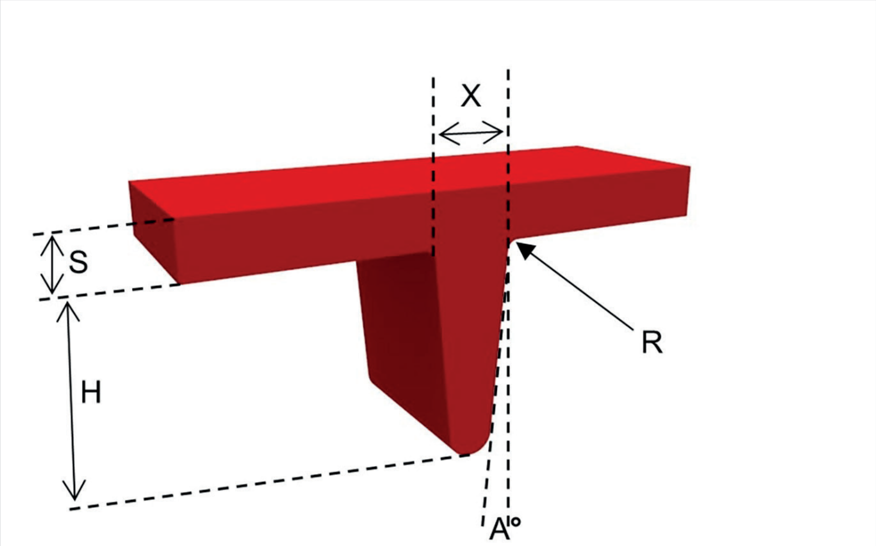
Where a rib meets a wall there should be a radius on both corners. Again, this will result in a thickening of the wall at this point. There are various options in such circumstances (see Design for Appearance).

Ribs

component without increasing the overall wall thickness and so increasing the weight of the component. However, ribs can increase the risk of warping and appearance problems. For successful rib design the following guidelines should be followed:

- To reduce sink marks on the surface the rib thickness should not exceed 50% of the adjoining wall thickness.
- To reduce stress, filling and ejection problems the height of the ribs should not exceed three times the adjoining wall thickness. When more strength is required more ribs are recommended rather than an increased height. A deeper rib may buckle under load plus it is difficult to machine into the mould and may cause the part to stick in the mould.
- A minimum radius of 25% of the adjoining wall thickness should be incorporated at the base of the ribs.
- Ribs are most effective when placed down the length of the area subjected to bending.
- Rib spacing should be at least twice the nominal wall thickness.
- A draft angle of at least 0.5 degrees on each side should be incorporated in order to facilitate release from the mould.

Rib Guidelines



Component section = S
Draft per Rib Side = A = 0.5° - 1.5°
Rib Height = H = < 5 x S (usually 2.5 – 3 x S) Radius = R = > 0.25 x S – 0.4 x S
Rib thickness = X = 0.4 x S – 0.8 x S
Rib spacing = 2 x S –3 x S

Support Ribs

May be used on reinforcement for corners, side walls or bosses. The following guidelines should be adhered to wherever possible:

- Thickness should be between 50% and 70% of component wall thickness.
- Minimum distance between faces of ribs should be twice component wall thickness.
- Minimum length of support rib face attached to component wall should be twice wall
- thickness.
- Generous radii should be incorporated at rib ends.
- Minimum of 0.5 degree draft should be incorporated.
- Minimum length of support rib face attached to a boss should be four times wall thickness.

Bosses

These are usually incorporated to facilitate mechanical assembly. They can be designed to accommodate self-tapping screws, push-in or moulded-in inserts or used for ultrasonic welding. Therefore, the boss may have to withstand a variety of forces – tension, torsion, compression, shear and flexing.

Design suggestions:

- Wall thickness of the boss should be 50% to 70% of the nominal wall. However, this may not be sufficient to withstand the stresses imposed by an insert but a thicker section can cause sink marks. Frequently, a compromise is required.
- Minimum radius of 25% of the wall thickness at the base of the boss is recommended. Further strength may be achieved through the use of support ribs.
- Strength can also be increased by attaching the boss to a nearby wall using a rib.

All of the above boss layouts will result in sink marks and/or voids.

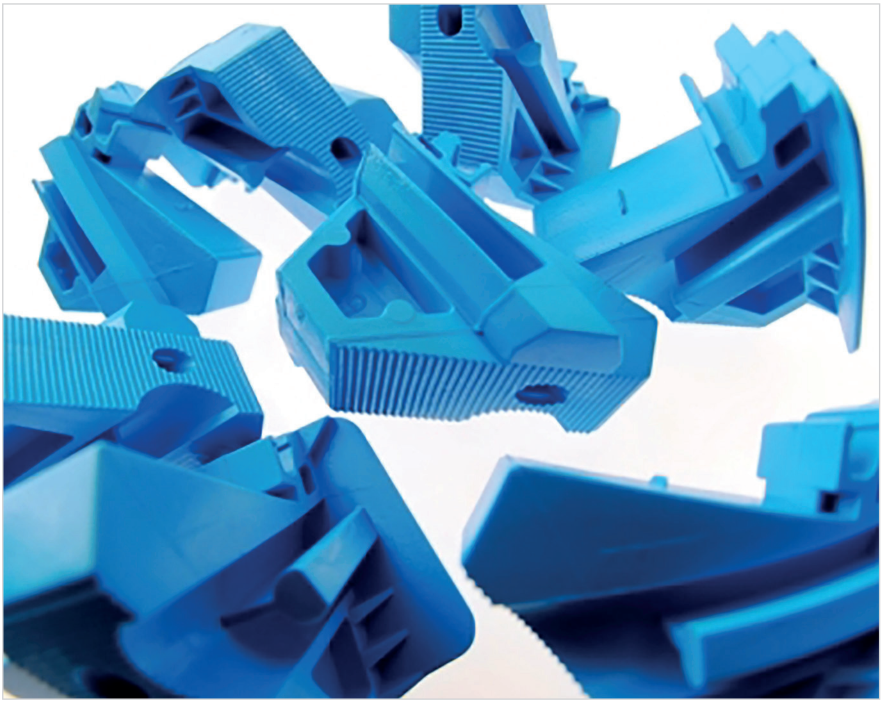
Typical layout for optimum strength and appearance.

Undercuts

Undercuts should be avoided if at all possible. Ideally mould tools should be straight open and close. In certain circumstances mouldings with slight undercuts can be ‘bumped’ off the core. The same results as an undercut can be achieved using certain design techniques. For some complex components, it may still be necessary to have side movements in the mould tool but it has to be accepted that this will increase tooling costs.

There are various alternative solutions that Rutland Plastics would be happy to discuss and advise on.

Coring



This refers to the elimination of material from a particular area of the component. It is a means of reducing wall thickness and material content. Imagine a 'Frisbee' – viewed from the top or the side and it appears a solid disc but viewed from below it is 'dish' shaped or 'cored out' in its most basic form.

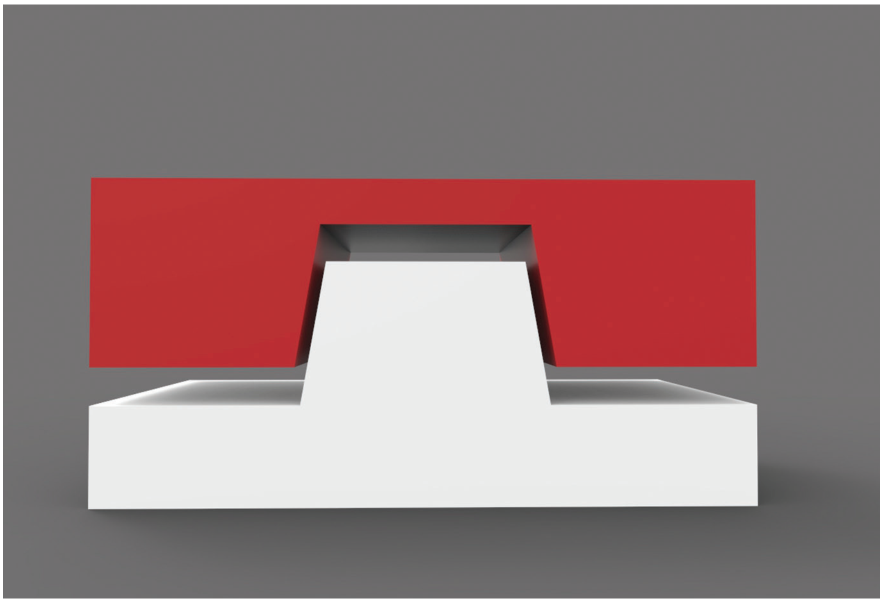
Coring can also refer to the addition of holes or vent slots in a component. Holes are associated with a number of problems as outlined elsewhere. Irregularly shaped slots incorporating sharp internal corners should be avoided. The weld or flow-lines created by holes may be aesthetically unacceptable.

To minimise these problems:

- Shortest distance between edges of any two holes or slots should be greater than twice nominal wall thickness.
- When positioning a hole or slot near the edge of a component the distance between the edge of a hole or slot and the edge of the component should exceed twice the nominal wall thickness.

Draft Angles

With a normal cavity and core the cavity is stationary and the core is attached to the moving platen. In the majority of cases the part 'sticks' to the core due to shrinkage of the material moving it away from the cavity and tightening it onto the core. In order to facilitate part removal from the mould it is necessary to incorporate draft angles. 0.25 degrees to 2 degrees per side for both inner and outer walls is generally sufficient. If the surface is polished the draft angle can be at the lower end of the scale, for textured surfaces more draft is required if 'scuffing' or sticking of the part in the mould is to be avoided. Cavities with texture require an additional 1° - 1 1/2 ° of draft per 0.025mm of texture depth. If there is texture on the core then even greater draft angles are required as the material tends to shrink around the core.



It should be remembered that even ribs, bosses, depressions and holes must have draft.

Textures and Lettering

Textures and lettering can often be moulded onto the surface of the part. Texturing can help in hiding surface imperfections.

Recesses

Recesses or depressions in the surface of the plastic do not usually present problems. However, they can impact on the flow of the plastic causing weld lines, flow marks and possibly holes in the surface due to trapped air. Recesses can be eliminated by reshaping the wall to maintain a uniform section but if this is not possible then problems can be minimised by using ample radii.

Design for Recylability

Environmental legislation is increasingly focusing on the recyclability of products. This is already in evidence in the automotive and consumer goods industries and is likely to become increasingly prevalent. Plastics are a particular target because of the disposal problems associated with them coupled with the fact that the vast majority of plastic materials can be relatively easily recycled (although cost is currently prohibitive in most cases).

Although functionality is still the primary consideration in design, designers now have a responsibility to design with economic use of materials and end of life recycling in mind. The following guidelines should be followed for products designed to be recyclable:

- Avoid using metal inserts as they are difficult to separate and make recycling uneconomic.
- Avoid using self-tapping screws and use snap-fit wherever practical.
- Avoid bonding with polyester and polyurethane based adhesives. If unavoidable then use 'break-out' facilities to make for easy separation of the bonded section. (Break-outs could also be used for removing metal parts if practicable).
- Where possible make components from the same material and grade. Where different materials are used clearly mark each part with material identification for ease of sorting.
- Avoid using decorative paints, lacquers and protective coatings.
- When printing or hot-foil decoration is used, an easy to remove secondary moulding can be used as the base.
- Reduce the amount of material. Gas assisted injection moulding (GAIM) can be used to hollow out thick sections. Another option is to reduce the density of the material through the use of chemical foaming agents.

It is not always possible to use a single material as different components within a product or sub- assembly may have to perform different functions. This can add to the cost of recycling as the parts need to be sorted with the different materials clearly identified and separated.

Certain materials are more compatible with each other than others, as shown in the chart below. The degree of compatibility varies so some are compatible in small proportions. This means that it may not always be essential to sort the different materials.Certain materials are more compatible with each other than others, as shown in the chart below. The degree of compatibility varies so some are compatible in small proportions. This means that it may not always be essential to sort the different materials.

		MINOR MATERIAL																		
		ABS	ASA	PA	PBT	PBT+PC	PC	PC+ABS	PC+PBT	PE	PET	PMMA	POM	PP	PPO	PPO+PS	PS	PVC	SAN	TPU
MINOR MATERIAL	ABS	√	√	O	√	√	√	√	√	O	O	√	O	O	O	O	O	√	√	√
	ASA	√	√	O	√	√	√	√	√	O	O	√	O	O	O	O	O	√	√	√
	PA	O	O	√	O	O	X	X	X	O	O	O	O	O	X	O	O	X	O	√
	PBT	√	√	O	√	√	√	√	√	O	O	O	O	O	O	O	O	X	√	O
	PBT+PC	√	√	O	√	√	√	√	√	O	O	O	X	O	O	O	O	X	√	√
	PC	√	√	X	√	√	√	√	√	O	√	√	X	O	O	O	O	X	√	O
	PC+ABS	√	√	O	√	√	√	√	√	O	√	√	O	O	O	O	O	X	√	√
	PC+PBT	√	√	X	√	√	√	√	√	O	√	√	O	O	O	O	O	X	√	√
	PE	X	X	O	X	X	O	X	X	√	X	X	X	√	X	O	X	O	X	O
	PET	√	√	O	√	√	√	√	√	O	√	O	O	O	O	O	O	X	√	O
	PMMA	√	√	O	O	O	√	√	√	O	O	√	X	O	O	O	O	O	√	O
	POM	O	O	O	O	O	X	X	X	O	O	X	√	O	O	O	O	X	O	O
	PP	X	X	O	X	X	X	X	X	O	X	X	X	√	X	O	X	O	X	O
	PPO	O	O	O	O	O	O	O	O	O	O	O	O	O	√	√	√	X	O	O
	PPO+PS	O	O	√	O	O	O	O	O	O	O	O	O	O	√	√	√	X	O	O
	PS	O	O	O	O	O	O	O	O	O	O	O	O	O	√	√	√	O	O	O
	PVC	√	√	X	X	X	X	X	X	O	X	√	X	O	X	O	O	√	√	√
	SAN	√	√	O	√	√	√	√	√	O	O	√	O	O	O	O	O	√	√	O
	TPU	√	√	√	O	√	√	√	√	O	√	√	√	O	O	O	O	√	√	√

√ = Good compatability   O = Limited compatability with low volumes   X = Incompatible

Design for Economy

There are two areas of consideration in designing for injection moulding:

- Design of the part to minimise production costs
- Design of the part to minimise tooling costs

Production Considerations

Injection moulding is a heat process and plastics are poor conductors of heat, that is to say it takes a relatively long time for plastic parts to cool down. With this in mind the aim should be to keep sections of parts as thin as possible, this will not only mean shorter moulding cycle times but also less material content.

Strength

If the part needs some strength this can be achieved by the addition of ribs rather than thickening of the section. With plastics, thicker sections do not necessarily mean stronger parts, in some instances beyond a certain thickness the part can become more brittle due to the resulting lack of flex.

Material

Another consideration is choice of material type and grade. Thermoplastics can range in price from 80p per kg for basic polyolefins up to £25 per kg and beyond for PEEK and other specialised polymers. By designing strength into a part it may be possible to use a cheaper material but sometimes the reverse can be true and advice should be sought from the moulder.

The price of polymers is also governed to a certain extent by the amount purchased so some benefit may be gained by using a grade already used by the moulder. Be guided by the moulder rather than sticking rigidly to a specified grade.

Eliminate Assembly

If a number of parts are required, and especially when converting from metal or a different production method, look at the possibilities of combining two or more parts into a single moulding so eliminating assembly. It is often possible to produce complex single mouldings that would not be possible with a different method of production. In a similar vein, where assembly is required every effort should be made to make this as simple as possible with the use of snap-fits, for example.

Tooling Considerations

Combining Parts

Injection mould tooling is relatively expensive but it is not always correct to assume that it is only suited to long production runs. If it is possible to combine a number of parts into a single moulding, for example, then the cost of assembly and possible other ancillary parts can be saved making shorter production runs economical.

Rutland Plastics were approached to replace a number of relatively low volume polyurethane foam mouldings produced by reaction injection moulding (RIM). These parts required finishing, including painting, as well as assembly operations. One of these parts was a base that contained a filter. The original part had two steel runners secured by 3 screws each to hold the filter, along with 2 aluminium baffles that were also attached by screws. The final injection moulding had moulded in runners and baffles so eliminating not just the costly parts but also the assembly operation. Although the resulting mould tool was complex and therefore expensive the payback was less than 2 years on just 400 bases per month.

Keep it simple

The simpler the mould tool the lower the cost. If possible avoid holes in side walls of parts, undercuts and other complex features. The mould tool can then be what is referred to as straight open and close. If the part does need side holes, for example, then these can either be moulded in which would require side movements in the mould tool, or a secondary operation will be necessary. Which approach is adopted will depend on anticipated quantities of the part – the higher the number of parts required the more economical it is to have the features produced by the mould tool rather than as a secondary operation.

The same rule applies to threads. Internal threads can of course be tapped afterwards but the usual method is to have them moulded in. This can be achieved in two ways, either with hand loaded cores or with auto unscrewing in the mould tool. The former requires operator intervention during the moulding process so adding to the part cost whereas the latter means the mould tool can be run fully automatically but will mean a higher tooling cost at the outset.

Multiples

To reduce both total tooling and part costs where a number of similar parts are required in equal quantities and in the same material, it may be possible to produce a family tool, i.e. all the parts moulded in a single cycle from a single tool. Also, where large quantities of a part are required, a multiple cavity tool could be used, i.e. two or more of the same part produced in a single cycle. Although the tooling cost would be higher than for a single impression tool, the part price will be lower so making it cost effective. The moulder can advise on the optimum configuration.

Conclusion

There are a number of factors that impact upon part and tooling cost: material selection, section of part, complexity of design, etc. What is certain is that the further down the design/production cycle you are the more costly any alterations become. Therefore, it is advisable to get the initial design right for injection moulding and to this end you should involve your injection moulder at the very early stages of the project to ensure correct design and material selection for the given application.

And Finally...

Consider all the factors and then establish the design. Rutland Plastics would be happy to help with this aspect. We can advise on suitability for injection moulding and cost-saving options if required. Rutland Plastics can take a sketch and prepare full drawings for you.

Once your requirements are known the material can be selected. Again Rutland Plastics can advise on the most cost-effective solution combining material properties and design features.

Critical dimensions, surface finishes, flatness, etc. must be specified along with realistic tolerances.

Involving Rutland Plastics at an early stage can allow concurrent engineering to take place. This will result in less wastage of time and resources in producing a successful design. It is possible to start this process with just a sketch rather than a full CAD design. When a plastic part is being designed to replace an existing product or component manufactured in another material it is advisable to start the design from scratch using critical dimensions and features rather than trying to replicate the existing item.

This design guide is just a starting point to help you understand how to get the best out of plastics. No obligation quotes can be provided to help you to assess the viability of your project.

If you have 3D CAD models then Rutland Plastics can accept most formats with STP the preferred choice although IGS and Parasolid are also acceptable. Remember, free advice is always available from Rutland Plastics via phone, fax, post or email, whichever method you prefer.

Acknowledgements

Plastic Part Design for Injection Moulding – Robert A. Malloy

Products Design for Injection Molding – Herbert Rees

GE Plastics Design Guide

Appendix 1

Answers to the following questions will aid the design process and help to ensure selection of the most appropriate material for the application:

Is the new part a replacement for an existing part or a completely new design?

....

If it is a replacement part can more than one existing part be incorporated into a single moulding?

....

Are there any special strength requirements? If so:

- Does it require impact strength?

....

- Does it require strength for loading?

....

- In either case, how much and is the impact/loading frequent or infrequent?

....

Is there likely to be exposure to chemicals? If so, specify the chemicals and whether exposure will be frequent or infrequent.

....

Is there likely to be exposure to UV light?

....

Will the part be exposed to extreme temperatures? If so, specify maximum/minimum temperatures and whether exposure will be frequent or infrequent.

....

What is the life expectancy of the part?

....

Are there any specific aesthetic requirements?

....

What colour(s) are required?

....

Is the part to be assembled with other parts? If so:

- What type of joints are required?

....

- Are the joints permanent or will there be a requirement for disassembly?

....

- If disassembly is required will this be frequent or infrequent?

....

Is there a requirement for the part to be recycled?

....

What are the anticipated annual quantities?

....

Our Accreditations

We hold a medical standard covering clean environment, non-sterile injection and flow moulded thermoplastic components, including machining, finishing and sub-assembly.

Quality assurance, control and monitoring procedures are built in to every aspect of our work.

bsi. ISO 9001 Quality Management | ISO 14001 Environmental Management | ISO 13485 Medical Devices Quality Management

INVESTORS IN PEOPLE

Cold Overton Road
Oakham
Rutland
LE15 6NU

E: enquiry@rutlandplastics.co.uk
T: +44 (0) 1572 723 476
W: rutlandplastics.co.uk

Office Hours
Mon-Thu: 08:00-17:00
Fri: 08:00-15:45

Company Registration Number: 560131
VAT Number: 330 3563 90