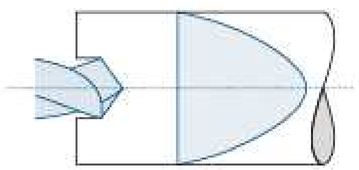
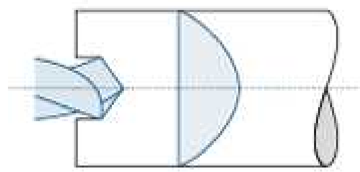
Drilling is a cutting process that uses a drill bit to cut a hole of circular cross-section in solid materials. The drill bit is usually a rotary cutting tool, often multipoint. The bit is pressed against the workpiece and rotated at rates from hundreds to thousands of revolutions per minute. This forces the cutting edge against the workpiece, cutting off chips (swarf) from the hole as it is drilled.

A specific procedure must be chosen for drilling into plastic parts, in order to eliminate component defects as far as possible.
When drilling, particular attention must be paid to the insulating characteristics of plastic. These can cause plastics (especially semi-crystalline ones) to quickly build up heat during the drilling process, particularly if the drilling depth is more than twice the diameter. This can result in drill "smearing" and inner elongation occurring in the material and can bring about compression stress in the component (particularly where drill holes are made in the cores of rod sections). The stress levels can be high enough to cause a high level of warping, dimensional inaccuracy, or even to or even cracks, fractures and bursting open of the finished component or blank. These effects can be avoided by choosing a machining method to suit the material.

Stress produced with a blunt drill





Stress produced with a sharp drill

For the above reasons, when drilling it is advisable to use coolants and also to frequently withdraw the drill bit to guarantee adequate cooling and chip removal. Drill holes can generally be produced using a well sharpened standard commercially available HSS drill bit. It is also advisable to use a drill bit with a reduced land width (synchronous drill) in order to reduce friction and consequently also heat accumulation. Manual feed should also be avoided to prevent the possibility of the drill becoming entangled and causing cracks.

Compared to unreinforced plastics, reinforced plastics have a higher residual tension level. The use of reinforcing materials and fillers also makes the products harder and more brittle. Impact strength also diminishes. Consequently, these products are particularly susceptible to cracks, particularly during the drilling process. During the machining process, the residual tensions can be released, culminating in effects ranging from high levels of warping to crack formation or complete rupture.
Consequently, the following notes should be taken into account during the machining process:

- The semi-finished products should be heated where possible prior to drilling up to 120°C (heating time ~1 hour per 10 mm of cross section)
- Carbide tipped or even better diamond tipped tools should be used for machining
- When tensioning and fixing the workpieces, pay attention to freedom from distortion, i.e. subject the material to the smallest possible bending, tensile or compressive forces.

Small diameters (0.5-25mm)
High-speed steel (HSS) drill bits are usually sufficient here. Twist drills with an angle of twist of between 12 and 16° and very smooth helical flutes are very suitable for this purpose and also help to ensure good chip removal. As mentioned above, ensure that the drill bit is withdrawn frequently (brief drilling periods) in order to improve the removal of chips and prevent the build-up of heat.
For drill holes in thin-walled workpieces, it is advisable to select a high cutting speed and where applicable a neutral (0°) rake angle. This will prevent the drill bit becoming entangled in the workpiece and tearing open the hole or lifting the workpiece.
Large diameters (25mm and bigger)
It is advisable when working with holes of this size to produce a pilot hole and then finish machine using an internal cutting tool / circle cutter. The pilot hole should not have a diameter greater than 25mm.
Holes drilled into rod sections should be produced from one side only, in order to avoid the unfavourable stress conditions which occur when holes drilled from two sides meet, which can result in the fractures in the rod wall.
In extreme cases, it can even be advisable for instance to heat the blank to around 120°C (heating time appr. 1 hour per 10 mm of cross section) and to carry out pilot drilling in this condition. To ensure dimensional accuracy, finish machining then takes place after the blank has cooled down completely.

Inside sharp corners same as fillet and radius:

Generously rounded corners provide a number of advantages. There is less stress concentration on the part and on the tool. Because of sharp corners, material flow is not smooth and tends to be difficult to fill, reduces tooling strength and causes stress concentration. Parts with radii and fillets are more economical and easier to produce, reduce chipping, simplify mold construction and add strength to molded part with good appearance.

Sharp Corners general design guidelines in injection molding suggest that corner radii should be at least one-half the wall thickness. It is recommended to avoid sharp corners and use generous fillets and radii whenever required. During injection molding, the molten plastic has to navigate turns or corners. Rounded corners will ease plastic flow, so engineers should generously radius the corners of all parts. In contrast, sharp inside corners result in molded-in stress particularly during the cooling process when the top of the part tries to shrink and the material pulls against the corners. Moreover, the first rule of plastic design i.e. uniform wall thickness will be obeyed. As the plastic goes around a well-proportioned corner, it will not be subjected to area increases and abrupt changes in direction. Cavity packing pressure stays consistent. This leads to a strong, dimensionally stable corner that will resist post-mold warpage.

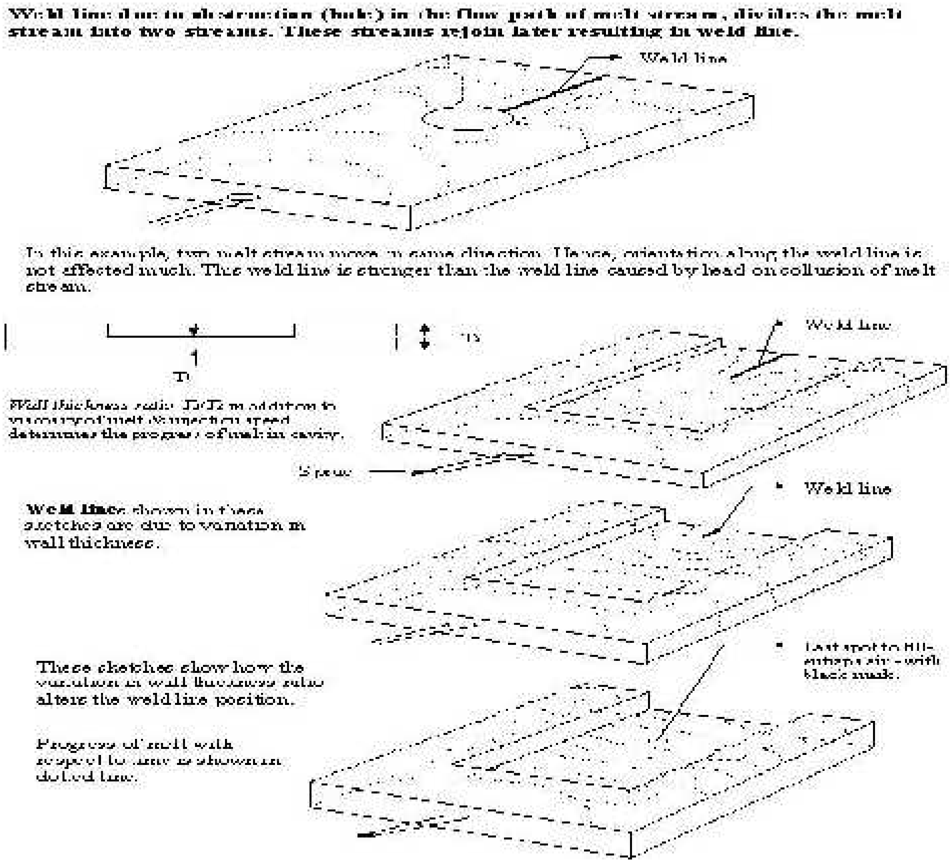
Sharp corners increase concentrations, which are prone to air entrapments, air voids, and sink marks hence weakening the structural integrity of the plastic part. It must be eliminated using radii whenever is possible. It is recommended that an inside radius be a minimum of one times the thickness. At corners, the suggested inside radius is 0.5 times the material thickness and the outside radius is 1.5 times the material thickness. A bigger radius should be used if part design allows

Weld lines:

In manufacturing, the Weld line or Knit line or Meld line is the line where two flow fronts meet when there is the inability of two or more flow fronts to "knit" together, or "weld", during the molding process. These lines usually occur around holes or obstructions and cause locally weak areas in the molded part. Knit lines are considered molding defects, and occur when the mold or/and material temperatures are set too low: thus the materials will be cold when they meet, so that they do not bond perfectly. This can cause a weak area in the part which can cause breakage when the part is under stress. There are many Computer Aided Engineering tools that are available that can predict where these areas could occur.

Knit lines could be caused by different causes:

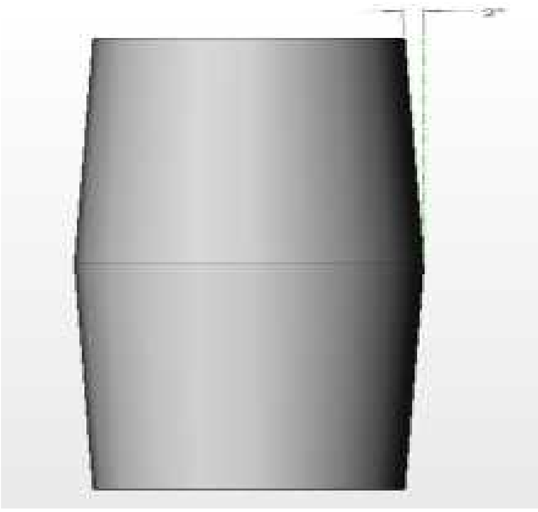
- Low temperature of molding machine barrel
- Inadequate back pressure
- Injection pressure or injection speed is too low
- Low mold temperature
- Small injection gates and/or runners
- Improper location of injection gate
- Excessive gate land length
- Improper flow rate of injected materials
- Inconsistent process cycle



Draft angles or Taper:

Draft angle design is an important factor when designing plastic parts. Because of shrinkage of plastic material, injection molded parts have a tendency to shrink onto a core. This creates higher contact pressure on the core surface and increases friction between the core and the part, thus making ejection of the part from the mold difficult. Hence, draft angles should be designed properly to assist in part ejection. This also reduces cycle time and improves productivity. Draft angles should be used on interior and exterior walls of the part along the pulling direction.

The minimum allowable draft angle is harder to quantify. Plastic material suppliers and molders are the authority on what is the lowest acceptable draft. In most instances, 1degree per side will be sufficient, but between 2 degree and 5 degree per side would be preferable. If the design is not compatible with 1 degree, then allow for 0.5 degree on each side. Even a small draft angle, such as 0.25 degree, is preferable to none at all.



Gate Location:

Each injection mold design must have a gate, or an opening that allows the molten plastic to be injected into the cavity of the mold. Gate type, design and location can have effects on the part such as part packing, gate removal or vestige, cosmetic appearance of the part, and part dimensions & warping.

To avoid problems from your gate location, below are some guidelines for choosing the proper gate location(s):

- Place gates at the heaviest cross section to allow for part packing and minimize voids & sink.
- Minimize obstructions in the flow path by placing gates away from cores & pins.
- Be sure that stress from the gate is in an area that will not affect part function or aesthetics.
    - If you are using a plastic with a high shrink grade, the part may shrink near the gate causing "gate pucker" if there is high molded-in stress at the gate
- Be sure to allow for easy manual or automatic degating.
- Gate should minimize flow path length to avoid cosmetic flow marks.
- In some cases, it may be necessary to add a second gate to properly fill the parts.
- If filling problems occur with thin walled parts, add flow channels or make wall thickness adjustments to correct the flow.

Gates vary in size and shape depending upon the type of plastic being molded and the size of the part. Large parts will require larger gates to provide a bigger flow of resin to shorten the mold time. Small gates have a better appearance but take longer time to mold or may need to have higher pressure to fill correctly.

Gate Types

There are two types of gates available for injection molding; manually trimmed and automatically trimmed gates.

Manually Trimmed Gates:

These type of gates require an operator to separate the aprts from the runners manually after each cycle. Manually trimmed gates are chosen for several reasons:

- The gate is too bulky to be automatically sheared by the machine
- Shear-sensitive materials such as PVC cannot be exposed to high shear rates
- Flow distribution for certain designs that require simultaneous flow distribution across a wide front

Automatically Trimmed Gates

These type of gates incorporate features in the tool to break or shear the gates when the tool opens to eject the part. Automatically trimmed gates are used for several reasons:

- Avoiding gate removal as a secondary operation, reducing cost
- Maintaining consistent cycle times for all parts
- Minimizing gate scars on parts

Undercuts:

In manufacturing, an **undercut** is a special type of recessed surface. In turning, it refers to a recess in a diameter. In machining, it refers to a recess in a corner. In molding, it refers to a feature that cannot be molded using only a single pull mold. In printed circuit board, construction it refers to the portion of the copper that is etched away under the photoresist. In welding, it refers to undesired melting and removal of metal near the weld bead.

Undercut - Any indentation or protrusion in a shape that will prevent its withdrawal from a one-piece mold.

Undercuts on molded parts are features that prevent the part from being directly ejected from an injection molding machine. They are categorized into *internal* and *external* undercuts, where external undercuts are on the exterior of the part and interior undercuts are on the inside of the part. Undercuts can still be molded, but require a *side action* or *side pull*. This is an extra part of the mold that moves separately from the two halves. These can increase the cost of the molded part due to an added 15 to 30% cost of the mold itself and added complexity of the molding machine.

If the size of the undercut is small enough and the material is flexible enough a side action is not always required. In these cases the undercut is stripped or snapped out of the mold. When this is done usually a stripping plate or ring is used instead of stripper pins so that the part is not damaged. This technique can be used on internal and external undercuts.