Injection Moulding

Among other 3D technologies our company offers batch or mass production of injection moulding parts via our trusted manufacturers offering guaranteed high quality of end-user components.

Below you can find a design guide with best practices and design tips can to assist you in the design engineering process and to create complex shapes while:

- Allowing plastic to flow easily and uniformly around the part.
- Allowing the plastic to cool quickly and evenly, resulting in a stable and accurate part.

These general tips will improve part quality, mould ability and cycle time based on known implementation and characteristics of the injection moulding process.

3Dtechnologies4U

Surface Finishing

We have the capability to provide excellent surface finishing offering some of the following options. Starting with bead blasting we remove the machining marks left to the parts followed by polishing. It is important to clarify that the more polishing required the higher would be the cost of the mould tooling and more time would be needed to complete the mould.

Finish Quality	Cost
Dull finish	Low
Satin finish	Low
Low polish	Low
Medium polish	Medium
High polish	High
Textured finish	Very high

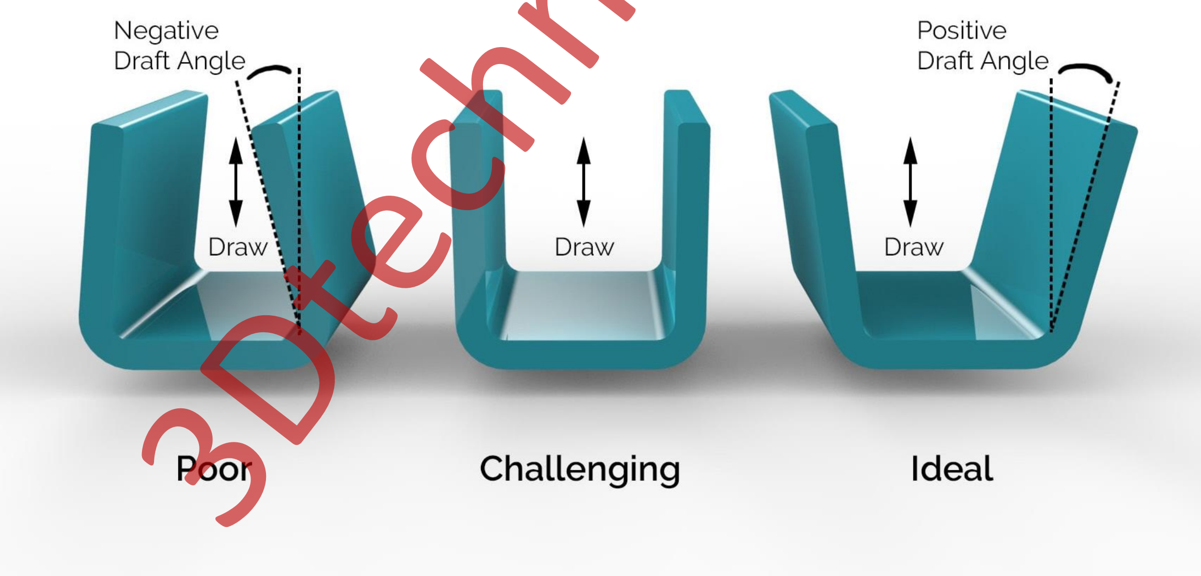
Draft Angles for Injection Moulding

It is known that CNC machining or 3D printing don’t require any draft angles and both technologies can manufacture parts with vertical walls and features. However, in injection moulding this is impossible and a drafted angle or slanted wall would be required. The reason is that components with vertical walls would not be easily removed from the mould while the part is cooling down and contracts onto the core.

Draft, or the application of a slight taper to every surface in the direction of pull on an injection-moulded part, is a small and even tedious design element — but one that's vital to the success of a project. To visualize draft, envision an ice cube tray: the slight taper allows ice cubes to slide out easily without falling victim to excessive suction or friction. Parts that are lacking the appropriate amount of draft — or a suitable draft substitute — will not properly eject from the mould.

Nowadays, all the CAD system have simplified operations to easily add draft angles on the part prior the design completion.

Feature Description	Minimum draft angle
Almost vertical	0.5º
Common geometries	2º
Shutoff surfaces	3º
Features with light textures	3º
Features with medium textures	5º



The minimum draft angle for any given part is largely driven by the depth of draw, the wall thickness, the material's shrink rate, and the surface finish or texture that is to be applied. As a general rule, a draft angle of 1.5 to 2 degrees is required for most parts, but draft should average about an additional degree for each extra inch of part depth. Note that if a part is

3Dtechnologies4U

very small, there's some more flexibility to decrease draft below 1.5 degrees. However, for most parts, 1.5 degrees is the minimum draft requirement.

That said, texture also plays an important role in determining draft. Many injection-moulded parts have a leather grain or other texture applied to their surface for aesthetic purposes; however, depending on how deep the texture is, the draft angle may need to be increased to ensure the texture won't be scraped off or damaged during the ejection process.

Undercuts in Injection Moulding

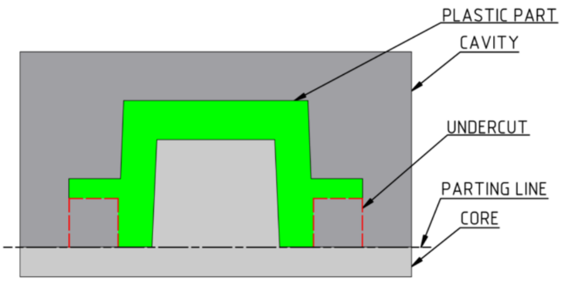
Complicated geometries and components that contain snap fittings will need a different manufacturing approach which needs to be considered by the design engineer. Undercuts are will need an alternative way of design and manufacturing simply because they will prevent the ejection of the component from the mould. In these instances, as undercuts can be found commonly, they can be formed by using a feature which is called 'slide' or 'side core'.

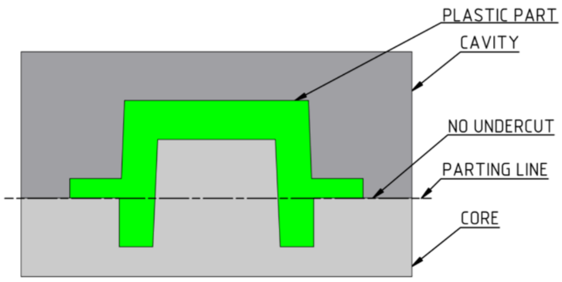
The position of the slide depends on the orientation and location of the undercut. Usually if the undercut sits on the external side of the part, then the side core feature will slide in from the side during the moulding process.

If the undercut is located at the same side of the component, then possibly depending on the geometrical features, more than one undercuts could be addressed by a single slide.

All the undercuts that exist in the inner side of the component could be similarly addressed by another sliding feature which called the 'lifter'. Due to the constraints of the manufacturing tooling and process the lifter and the sliders shall have a limited depth or pull characteristics.

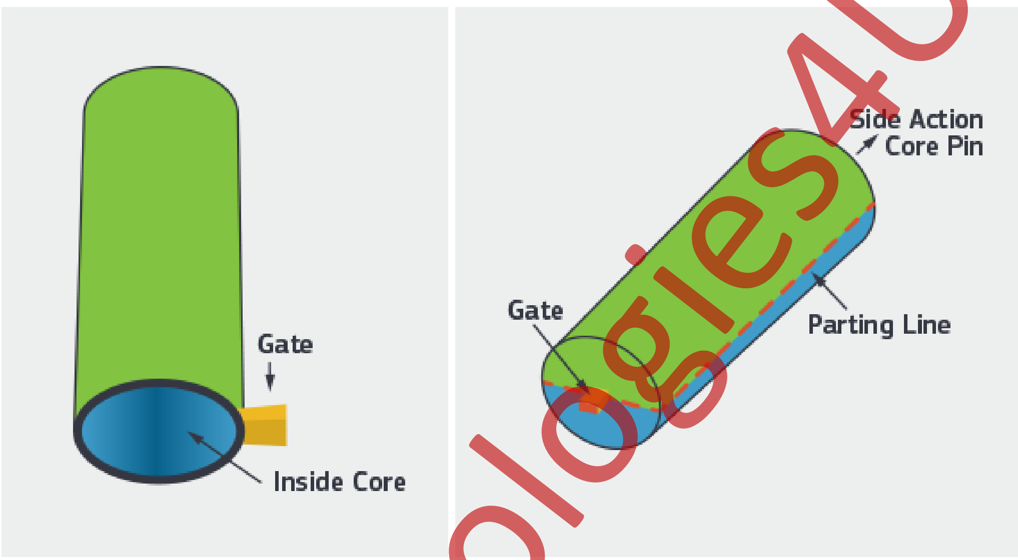
3Dtechnologies4U





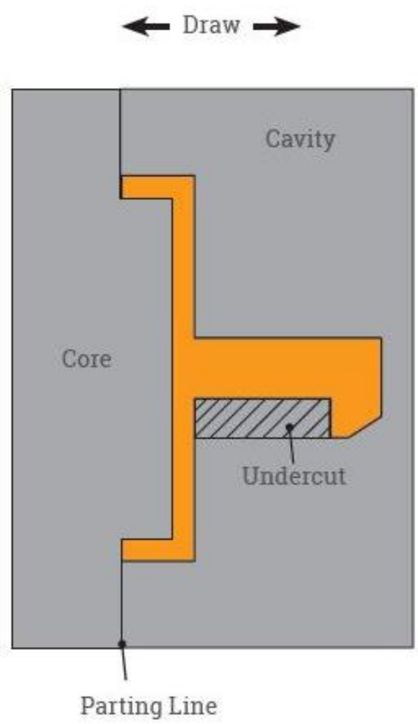
Side-Actions

A perpendicular side-action is ideal for cylindrical parts, as the mould is split horizontally along the part. After the resin is shot into the mould and begins to cool, the side-action slides on an angled pin until its clear from the undercut, allowing it to be freely ejected.



Sliding Shutoffs

This technique uses create clip- and hook-style components to lock together two halves of a mould. During mould operation, these mechanisms seal together, "shutting off" certain areas of the part to create complex features, such as holes.



3Dtechnologies4U

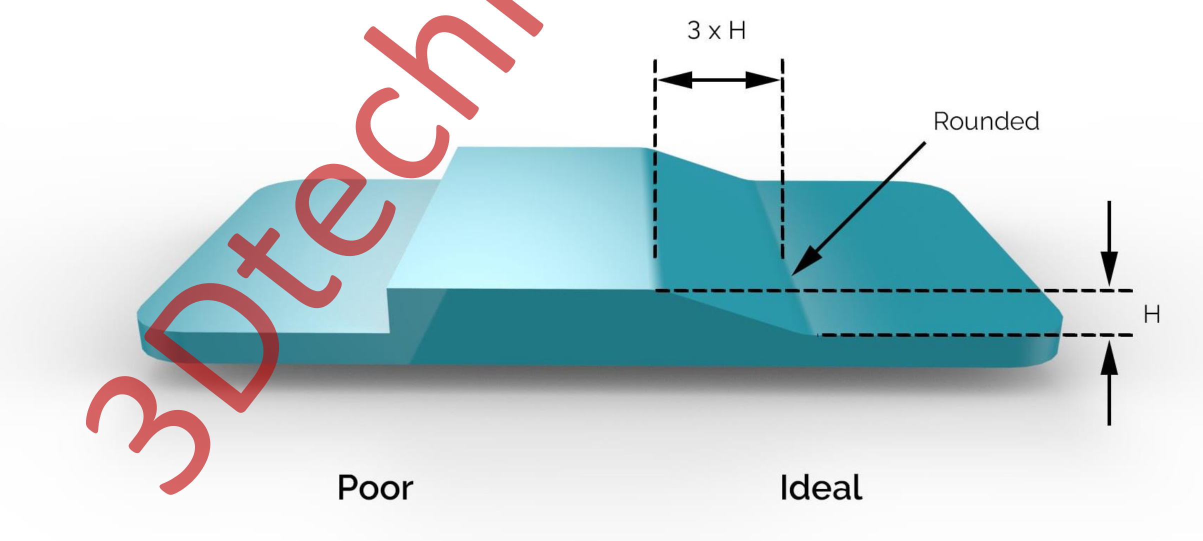
Wall Thickness

If you take apart any of the plastic appliances around your home (as most engineers probably did as children) you'll notice that the walls for most parts are about 1 mm to 4 mm thick (the optimal thickness for moulding), and uniform for the entire piece. Why? Two reasons

First of all, thinner walls cool faster, shortening the cycle time of the mould, the amount of time it takes to make each part. If a plastic part can cool faster after the mould is filled, then it can safely be ejected sooner without warping, and because time on the injection machine costs money, the part is less expensive to produce.

The second reason is uniformity: In the cooling cycle, the outer surface of a plastic part cools first. Cooling causes contraction; if the part is of uniform thickness, then the entire part will shrink away from the mould uniformly as it cools, and the part comes out smoothly.

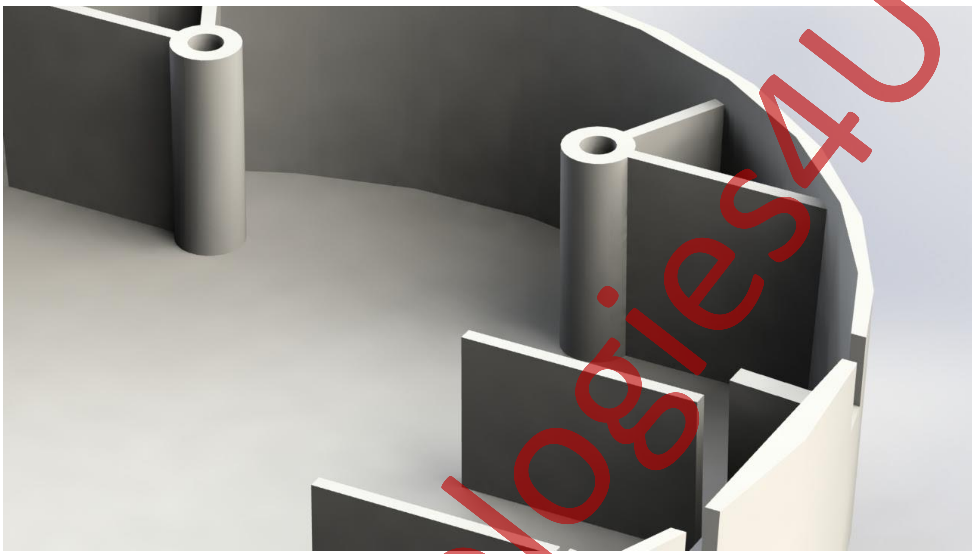
However, if the part has thick and thin sections next to each other, then the molten centre of the thicker area will continue to cool and contract after the thin areas and surfaces have already solidified. As this thick area continues cooling, it keeps contracting, and it can only pull material from the surface. The result is a little dimple on the surface of the part called a sink mark.



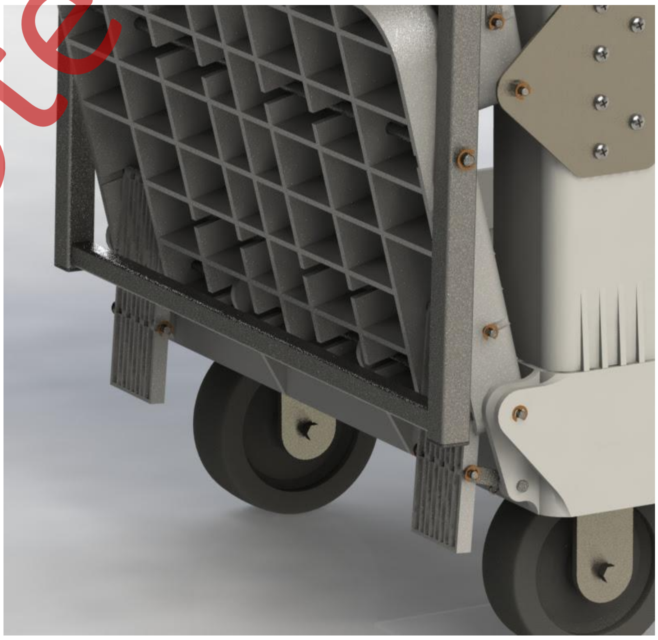
You may be able to address thick walls issue with simple solutions and design tips as follows: The first thing to do is to notice the areas which are a problem. In the part below, you can see two common issues: thickness around screw holes, and thickness where strength is needed in the part.

3Dtechnologies4U

For screw holes in an injection moulded part, the solution is to use “screw bosses”: a small cylinder of material directly around the screw hole, tied to the rest of the housing using a rib or a flange of material. This allows for more uniform wall thickness and fewer sink marks.



When an area of the part needs to be especially strong, but the wall is too thick, the solution is similarly simple: ribbing. Instead of making the entire part thick and difficult to cool, thin the exterior face into a shell, then add vertical ribs of material to the interior for strength and rigidity. In addition to easier moulding, this reduces the amount of required material, reducing costs.

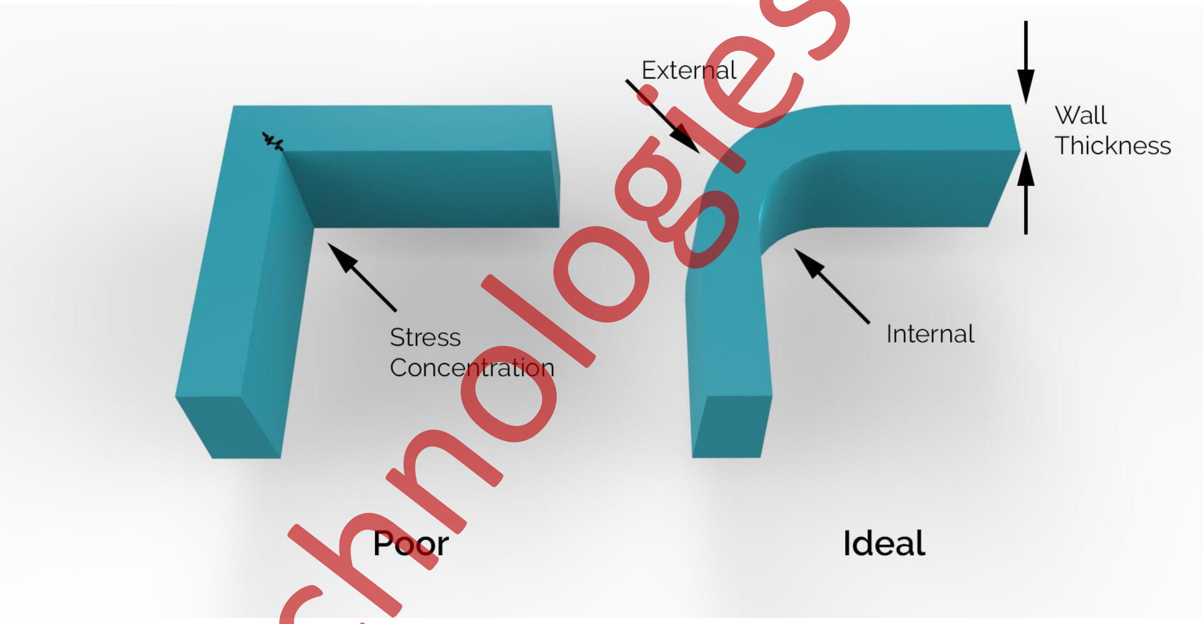


3Dtechnologies4U

Radius and Corners of your parts

Correct placement of corner radii in injection moulding design creates strong, high-quality and cost-effective plastic parts. Sharp external corners are ok and sometimes necessary to fulfil product requirements, such as triangular shaped items. However sharp corners can present challenges when designing for injection moulding, as they can cause stress resulting in a poor product, radii are key to reducing stress.

There are two types of radiuses, internal and external. Internal edges should be rounded to a minimum of 0.5 times the wall thickness. External edges should be rounded to a minimum of 1.5 times the wall thickness.



3Dtechnologies4U