Dienamics
Ideas Moulded by Experience

Injection Moulding Product Design Checklist

This is handy checklist will help your product design become Injection moulding ready.

When you go to design your plastic part think of the golden rule of Injection Moulds.

Where the isn't plastic, there must be steel.

☐ **My product has no undercuts.**
If there are undercuts (parts in your design that can't be formed by either the fixed or moving half of the mould) we can make sliding, sprung or collapsible cores. This does however, add to your tooling cost and can limit the number of the cavities in your tool. A product that can be formed completely by both halves of the tool is what we call "Open-and -Shut". This means that there is no other movement needed to form the tool. All the mould is doing is opening and shutting.

☐ **My product is a consistent thickness.**
We can mould certain plastics to thicker wall sections, but we usually find that 2-4mm thick walls have the best balance when it comes to strength, cooling time, and things like UV Stabilization, Flame retardancy and plastic flow in the mould. If there must be different thickness of wall section, try to make the transition between them as smooth as possible. Longer tapering wall section are the best for wall thickness transitions, but if that can't be achieved at least radius the corners.

☐ **My product has adequate draft.**
Once the mould has started to split, you want the product to be clear of the mould surface, so that the product can eject cleanly. If we had perfectly flat surfaces, the mould would have to push the part off the entire length of the mould and the walls of the mould would scuff the entire length of the plastic. Instead, we add a slight draft angle so that once the product has been lifted off its surface, it's free to fall away from the mould with no scuffing.

☐ **My product has no knife edge corners.**
Thin walls or edges of steel that come down to an acute angle cause problems. This is for a couple of reasons. First, they are difficult to machine. Thin walls of metal are prone to breaking. In external corners, the corners will chip of during machining or are broken in the mould. Sharp internal corners are hard to machine traditionally. All cutters have a radius (a Ø1mm cutter bit will leave a R0.5mm) and this means that any internal corner will have a small radius at it's very tip. Thin sections of steel also tend to super-heat and end up damaging your product's appearance. Try to keep the steel at least 3mm thick.

☐ **My product is one consistent shape.**

In injection moulding, we inject hot, molten plastic into a mould under pressure. It wants to move through and fill the cavity as quickly as possible. If the plastic has to split into 2 flow fronts, to go around a certain part of steel or part of the mould, when the 2 flow fronts meet up again, the plastic will not bond together as well as before. This can be seen in plastics that have a metallic tint, as flow lines or weld lines, where the plastic has split and come back together. We can make the plastic go around parts like this, but when they meet again, there may be a line that is slightly visible, and it will become a weaker spot.