

JOHN D. GILL, AN ENGINEER FROM CAPPLUGS, EXPLAINS FIVE THINGS MANUFACTURERS NEED TO CONSIDER WHEN DESIGNING PLASTIC PARTS.

Injection moulding is the most common method of producing plastic components. When designing plastic parts, it is really important to understand what you can do in order to ensure the part can be manufactured as efficiently as possible.

A badly designed plastic part is difficult to produce, which means there may be quality issues with the part. As a result, this will mean it is expensive, both in the production of the part and the ongoing life of the part. The expense and difficulty can be avoided though by considering the following factors when designing the part.



Expense and difficulty can be avoided by considering the following factors when designing the part.



KEY DESIGN CONSIDERATIONS FOR PLASTIC PARTS

1 THICKNESS

Plastic parts can have a much thinner wall thickness than most designers realise. A small cap approximately 1" (25mm) diameter can have a wall thickness of just 0.040" (1mm) and function well as a protection cap or plug.

This thin wall means the parts can be produced quickly, maintain their shape and size during the production process, and have a short cycle time in the injection moulding machine. All of these elements ensure you receive a part that is consistently made to drawing and keeps production costs to a minimum.

2 UNIFORM THICKNESS

It is crucial to ensure that you design a part with a uniform thickness throughout. Parts with areas where the material is much thicker than the rest of the part are prone to becoming misshapen. If a plastic part cannot cool evenly at the same rate it is liable to become twisted or have areas that become dish shaped as they cool. All of this means that the dimensions of the part will not be consistent.

3 RADII

Plastic has to flow in and around the tool in order to produce the part, so it is crucial to include a radius on every sharp corner. Where you have a corner in the design, make the internal radius approximately half the wall thickness and ensure the outside radius is one and a half times the wall thickness, in order to maintain a consistent wall thickness around the corner.

5 TEXT

Having a name or logo on the part is a great way to advertise or to provide the part number for the part. This is also easy to do on plastic parts. Generally, the text will need to be raised on the part and a final height of 0.020" (0.5mm) is a good height. Fonts such as Century Gothic, Verdana or Arial are recommended.

4 DRAFT

The part has to come out of the tool efficiently and cleanly, so it is important to add a draft angle to faces to allow the tool to eject the part efficiently. A draft on the part will ensure that you do not see deep ejector pin marks on the part or marks on the part where it has been forced out of the tool. As a general guideline, aim for one to two degrees of draft when designing the part. Don't forget to maintain the wall thickness when adding the draft angle.

A plastic injection moulding company who designs their own tools will be able to help you with the design of your part, but thinking about the elements mentioned above will help both you and the company to get off to a smart start.