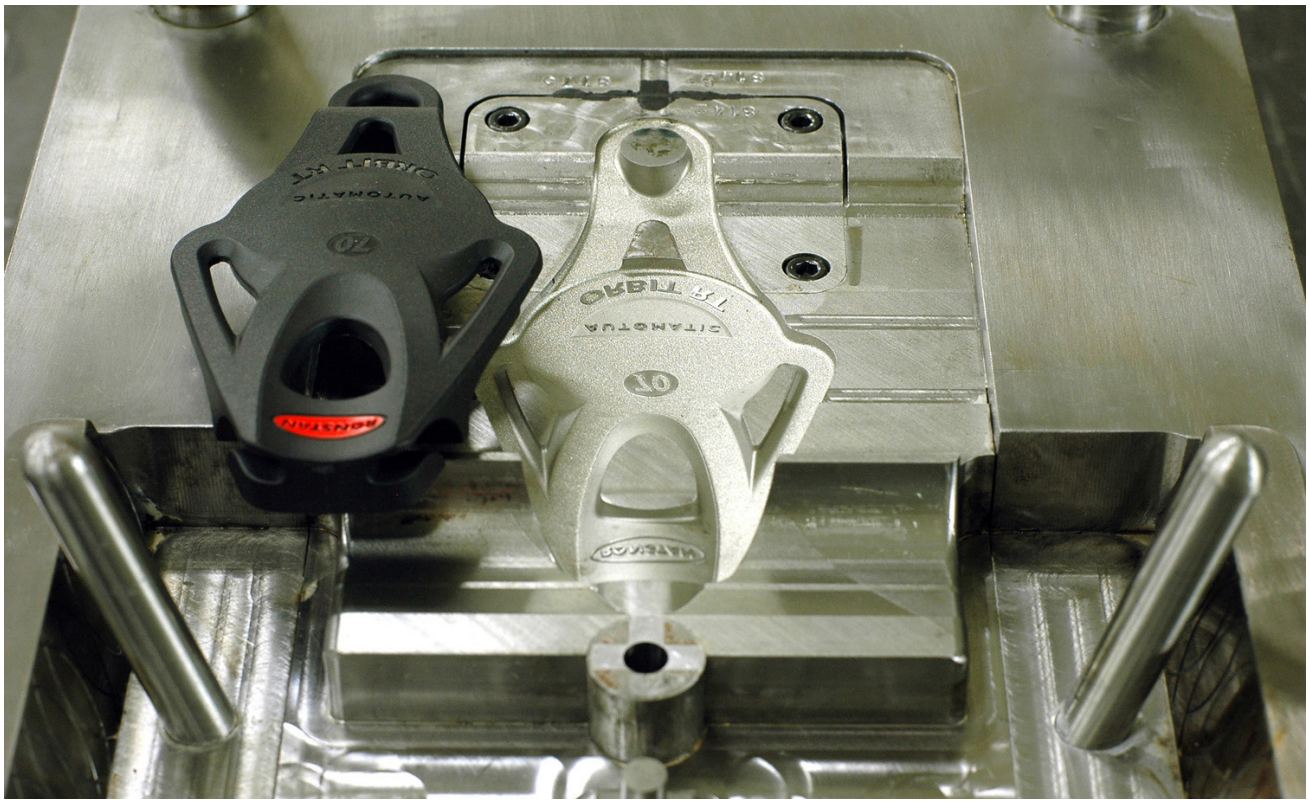
FORMERO

Design for Manufacturing

WHITE PAPER
Vol. 1

FORMERO

In this design guide we set out the basic guidelines for plastic injection moulded parts. Each product application is different, we recommend a thorough engineering analysis of your design by prototyping and testing new designs under actual use before committing to tooling.



Defining plastic part requirements

Ascertaining your part and material requirements will influence both part design, material selection and type of injection mould tool. Consider the following:

- Mechanical loads of your part design
- Temperature, chemical exposure, electrical, radiation or weather resistance
- Aesthetic requirements, such as transparency or colour, as well as identifying visual & non-visual areas
- Key dimensional tolerances
- Production quantities to determine mould design, tool automation & steel grade

Reducing your manufacturing costs

We identify manufacturing issues so you can achieve:

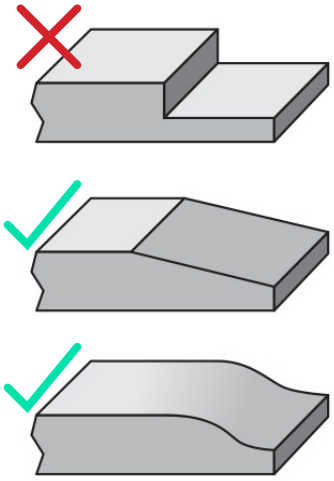
- Cost reductions
- Optimised part design
- Prevention of delays for time-sensitive products
- Streamline scale-up
- Cosmetic & structural requirements

FORMERO

Wall Thickness

Wall thickness influences many key characteristics, including mechanical performance, cosmetic appearance, mould ability and cost.

Too thick and you may have unsightly sink marks, warp or internal voids. Too thin and the molten plastic may harden before filling the mould (i.e. voids in the part). If different thicknesses are required, then make the transition as smooth as possible. All of this ensures the material can flow as easily and consistently within the mould.

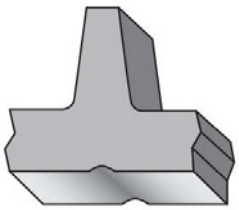


Gradual transitions between thick and thin sections.

Material	Recommended Thickness
ABS	1.1 – 3.5 mm
Acetal (POM)	0.6 – 3.8 mm
Liquid Crystal Polymer	0.75 – 3.0 mm
Long-Fibre Reinforced Plastics	1.9 – 25.4 mm
Nylon (PA 6)	0.75 – 3.0 mm
Polycarbonate (PC)	1.0 – 4.0 mm
Polyester	0.6 – 3.2 mm
Polyethylene (PE)	0.75 – 5.0 mm
Polyethylene Sulfide	0.5 – 4.5 mm
Polypropylene (PP)	0.6 – 3.8 mm
Polystyrene (PS)	0.9 – 4.0 mm
Polyurethane (PUR)	2.0 – 20 mm
PC/ABS	1.2 – 3.5 mm
Silicon	1.0 – 10 mm

Tips

- Keep wall thickness as uniform as possible
- Use gradual transitions between thick & thin sections
- Keep corner wall thickness as close as possible to the nominal wall thickness
- Thin, deep ribs are made using an EDM process, which adds cost to the mould

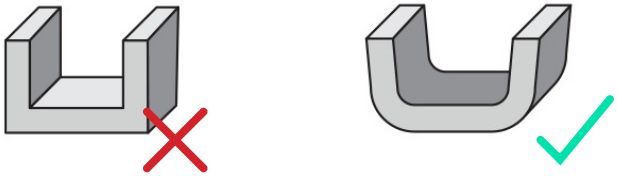


Sink mark caused by thick rib.

FORMERO

Radii - Rounded Corners

Sharp corners should be avoided because it prevents clean ejection of the part from the mould. Sharp internal corners tend to stick in the mould as the part shrinks onto the core. By adding radii, resin flows more easily within the mould and reduces part stress and aesthetic problems.



Ribs

Ribs provide a means to economically enhance stiffness and strength. However, the rib thickness and location can cause sink marks. Rib thickness and quantity also determines the cooling rate and degree of part shrinkage, which in turn can cause parts to warp. Consider the following to improve the finish of your parts:

- Reduce the thickness of the rib relative to the primary wall
- Add radius to the base of the rib
- Add additional or thinner ribs to off-set the flow of plastic

Carefully consider the location and quantity of ribs, as this may lead to parts warping or impact on the function of your part.

To prevent parts warping additional cooling in the mould can be applied.

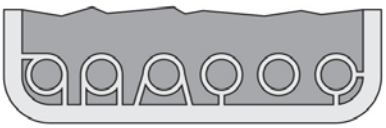
Tips

- A radius-to-thickness ratio of approx. 0.15 provides a good compromise between performance & appearance for most applications subjected to light to moderate loads.
- Minor blemishes on the outer surface created by ribs can be disguised by adding texture.

FORMERO

Bosses

Bosses are one of the most common features seen in plastic parts. However, similar to ribs if bosses get too thick relative to the nominal wall thickness, sink marks can occur. A boss-rib combination can eliminate sink marks. By using ribs to connect the boss to a side wall, this method of part design will provide the strength necessary to support screws,inserts, etc. Also, adding small radii to break the sharp corners will also greatly reduce stress concentrations.



Connecting bosses to wall without creating sink marks

Tips

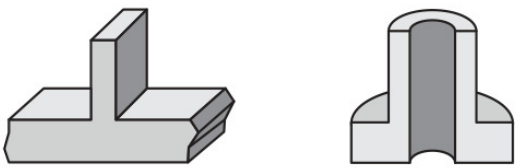
- Add appropriate draft on the outer diameter of a boss to help easy ejection from the mould
- A boss with a chamfer on top is a good lead for fasteners
- Thick bosses should be cored

Draft

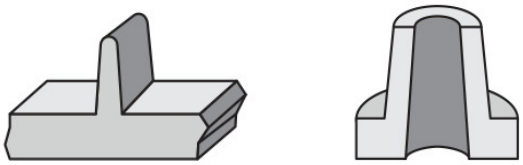
Draft is providing angles or tapers on product features (i.e. walls, ribs, posts & bosses). How a specific feature is formed in the mould determines the type of draft needed. As a general rule:

- Draft all surfaces parallel to the direction of the steel separation
- Angle walls and other features that's created in both mould halves to enable part ejection and maintain uniform wall thickness.

Examples (below) illustrate designs featuring no draft:



Examples (below) illustrate designs featuring draft:



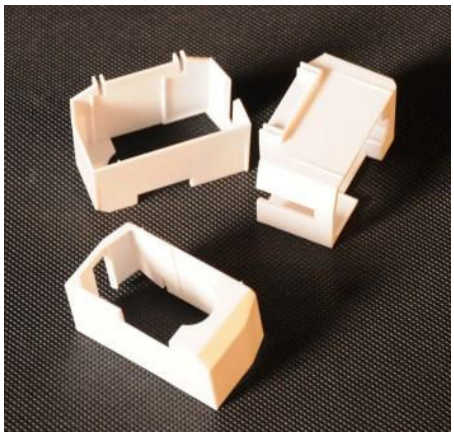
Tips

Adding draft angles to your design will:

- Minimise warping
- Increase the tool life
- Improve the surface finish on your plastic parts
- Improve moulding cycle time

FORMERO

Undercuts

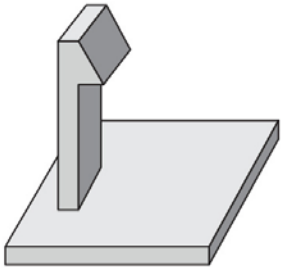


Undercuts are those features in an injection moulded part that prevent its ejection from the mould. Additional mechanisms in the mould need to be created to assist ejection, including slides, collapsible cores, lifters, etc. All these mechanisms add to the mould complexity and cost.

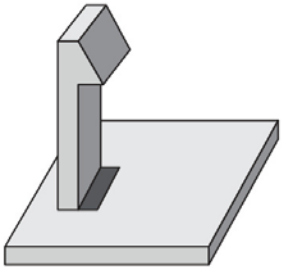
Avoiding undercuts maybe be possible, based on:

- Location of the parting line. However, mould orientation and parting line placement is dictated by part geometry,material flow, and a host of other factors
- “Bump-off”, which is a small undercut that can be safely removed from a straight-pull without the use of side actions. Only works for flexible materials
- Shut-off, which is a useful trick to deal with undercuts on internal regions of the part (for snap-fits) or on the sides of the part (for holes or handles)

Adding through holes (when the design allows for it) can give access to the underside of features that would otherwise be undercuts, simple modifications enable the mould to form a hole in the sidewall rather than with a side-action mechanism.



Side action required



No side action - through hole eliminates undercut

Tips

- Some undercut features can be successfully stripped from the mould during ejection if the undercut % is within the guidelines for the material type
- It’s not always possible to eliminate undercuts
- Our engineers will be able to assist you to provide the best outcomes based on budget, time frames, performance and appearance

FORMERO

www.formero.com.au

30-32 Ceylon Street,
Nunawading VIC 3131

Ph: +61 3 8899 7280

sales@formero.com.au