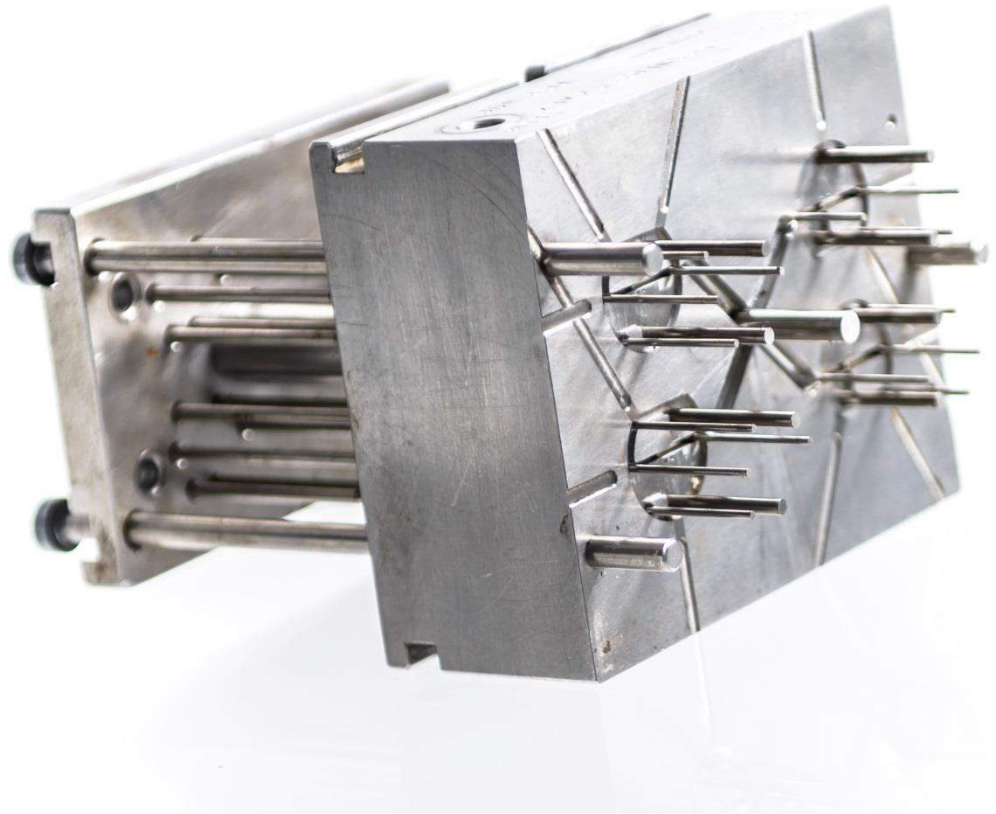
PENTAGON PLASTICS LTD
moulding Britain since 1972

DESIGN TIPS 2015



PENTAGON PLASTICS LTD

Unit 4 Blatchford Road

Horsham

West Sussex

RH13 5QR

Tel: 01403 264397

Pentagonplastics.co.uk

From concept and development through to tooling and the repeat production of your bespoke product

A Guidance Document for Designers of All Abilities

What is Plastic Injection Moulding?

Injection Moulding is a manufacturing process for producing parts from both thermoplastic and thermosetting plastic materials. Material is fed into a heated barrel, mixed, and forced into a mould cavity where it cools and hardens to the configuration of the cavity. After a product is designed, usually by an industrial designer or an engineer, moulds are made by a mould maker (or toolmaker) from metal, usually either steel or aluminium, and precision machined to form the features of the desired part. Injection moulding is widely used for manufacturing a variety of parts, from the smallest component to entire body panels of cars.

What does Pentagon Plastics Offer as a Service?

Pentagon Plastics is a privately owned ISO9001 accredited Plastic Injection Moulder of bespoke technical thermoplastic products. Pentagon offers one site of manufacture in its comprehensive Injection Mould Shop which is supported by its Rapid Prototyping and Toolmaking services.

Design Assistance services include the following: -

Plastic Material Selection
Feed Position
Optimum Material for Tooling
No of cavities based on Volumes
Evaluation of Product Design
Method of extraction from the Tool
Type of Tooling Configuration
Suitable Packaging

Design Tips Document

The product design is key to the Injection Moulding process for two main reasons:

- The part has to function as required for the intended application.

- It has to be possible to design a mould tool that will produce the part within the budget for the project.

The two points detailed above are however only the starting point of the elements that a designer needs to take into consideration. The information we have detailed is intended as a guide to highlight some of the key areas but the strong recommendation is to design out cost wherever possible.

This document will look at the following areas to hopefully assist in the design process:

1. General Design Tips

- ***Uniform Wall Thickness***
  - Thin Wall
  - Thick Wall
  - Rib to Wall
  - Warpage
  - Sink
- ***Coring Out***
  - When Required
  - Methods of Achieving
- ***Surface Finish***
  - Types Available
  - Draft Required
  - Advantages / Disadvantages

2. Product Design Tips in Relation to Tooling Considerations

- ***Parting Line***
  - What it is
  - Drafting
  - Open & Shut Tooling (lower cost)
  - More than one parting line (side movement, expensive)
  - Through Holes
- ***Core & Cavity***
  - What Are They
  - What Governs Which Side is Which (ejection)
  - Solid Core /Cavity or Split (cost, venting, polishing considerations
- ***Feed***
  - What is a Sprue / Runner / Gate
  - Types of Feed (Hot, Cold, Direct, Edge, etc)
  - Consideration of Aesthetics
- ***Ejection***
  - What it is
  - Types (Pin, Sleeve, Stripper, Up / Away, Up / In, Bump Off, etc)

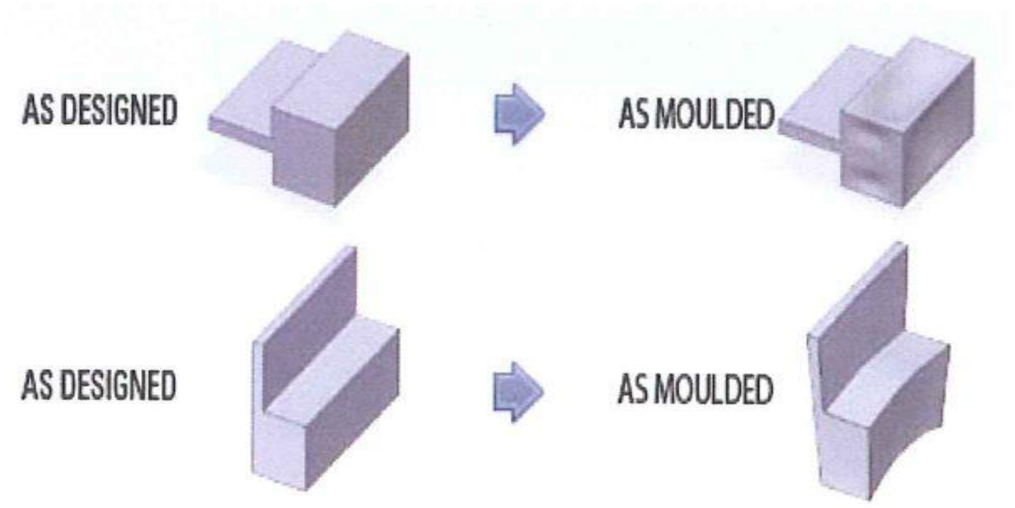
3. Material Considerations

- ***Importance of Deciding Early (Shrinkage & Tool Material)***
- ***Properties to Consider (Mechanical, Thermal, Tensile, Impact, Cost, etc)***
- ***List of Materials (Guide Table)***

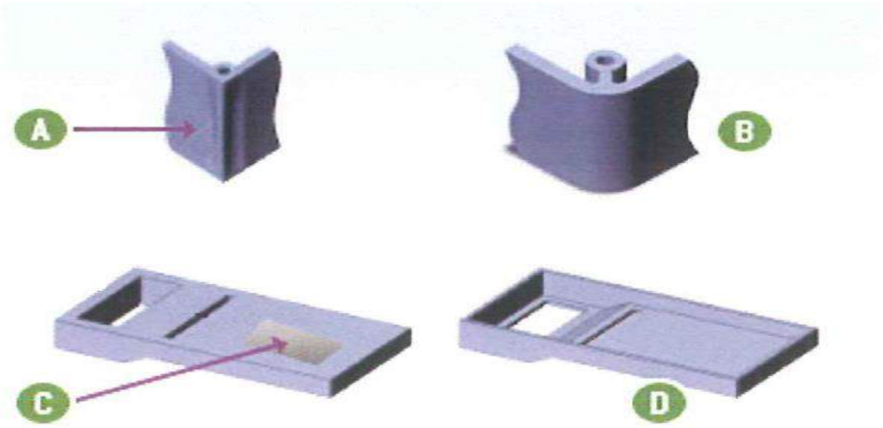
SECTION 1 - GENERAL DESIGN TIPS

Uniform Wall Thickness

Good injection moulded part design relies on consistent wall thickness to minimise the potential for warped or distorted parts. As the plastic solidifies in the mould it freezes from the outside toward the inside. In thick sections this results in inward pulling stresses due to contraction that can cause sink marks on the outer surfaces of the part. In addition,

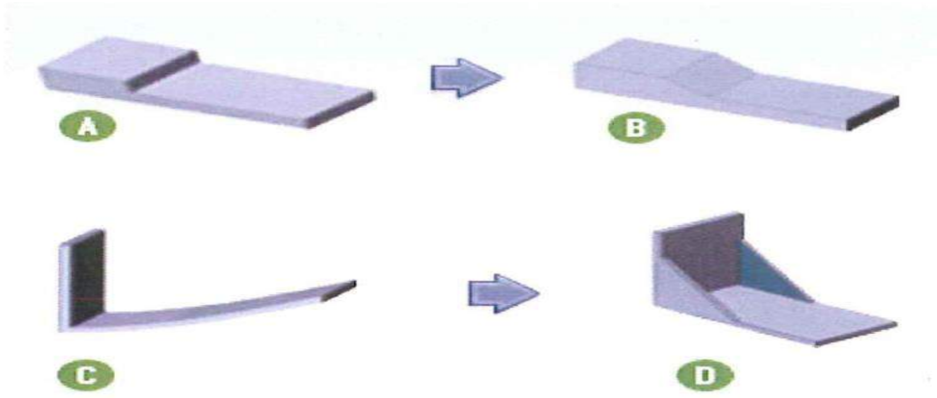


because thinner sections will freeze faster than thicker sections there is also the possibility of stresses building up between thick & thin sections resulting in warpage.



The use of thinner, uniform wall thicknesses helps to avoid sink. **A** = Boss in corner causes sink. **B** = Thinner walls on boss eliminates sink. **C** = Thick walls cause sink, warp & excess shrink. **D** = Thinner walls give accurate part.

Warpage due to stresses in step transitions between wall thicknesses can be improved through the use of a ramp and gussets can be helpful to provide support in corners.



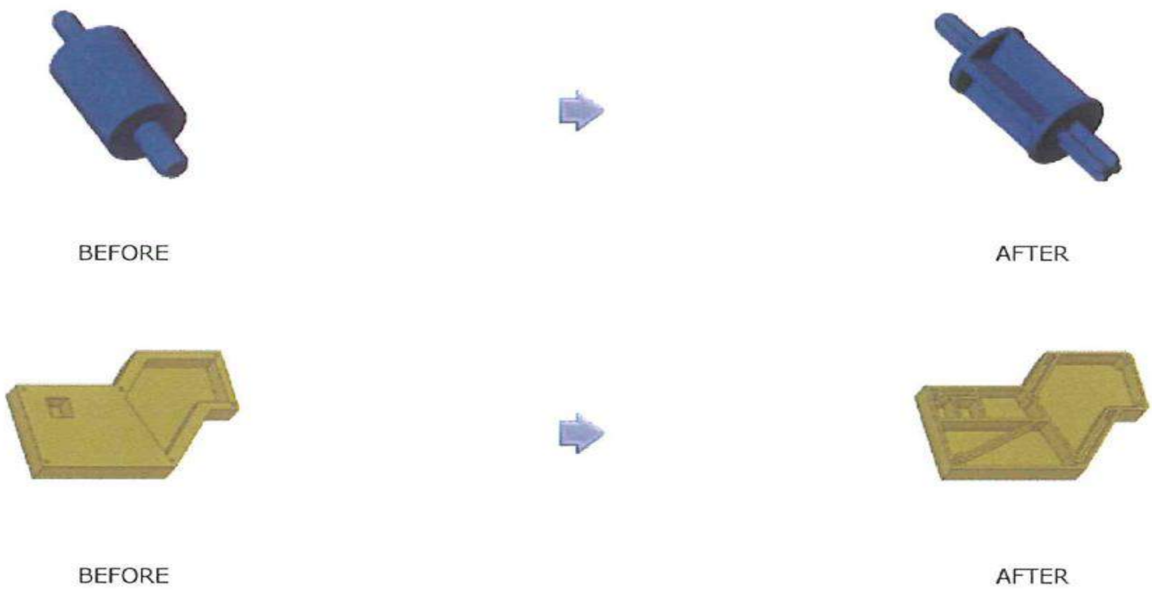
A = High stress concentrations. **B** = Reduced stress concentrations. **C** = Thinner walls result in shrinkage during cooling. **D** = Gussets provide additional support.

Thin ribs on thicker walls walls may provide stiffness but also can result in sinking on the outside of the wall. As a rule of thumb, the thickness of the rib should be about half of that of the wall to prevent this.

Coring Out

Plastic moulded parts which are too thick or have thick sections can be subject to excess shrink, sink, internal bubbles, voids and poor cosmetics. This is due to the fact that plastic shrinks as it cools from the molten injected state to solid room temperature parts.

Coring out is a technique where material is removed from a part leaving distinct walls and ribs which provide enough strength and mating surfaces for other parts in the assembly. Coring out is necessary to make the part mouldable and also saves cost and weight. The key points are leaving ribs in the right location and size to maintain strength and retaining features that interface with others.



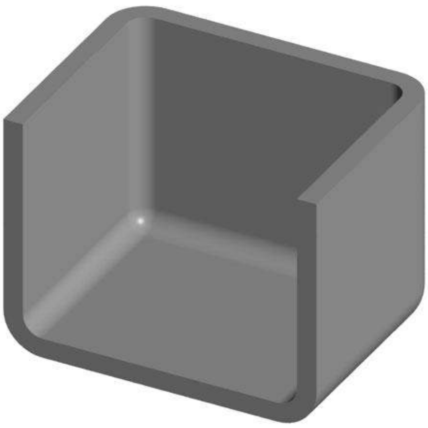
Surface Finish

Surface finish on a plastic part can serve many functions, from improving grip to hiding fingerprints to facilitating paint adhesion. Textures and lettering can be moulded on the surfaces as an aesthetic aid or for incorporating identifying information for end users. Texturing also helps hide surface defects such as knit lines and other surface imperfections. The depth of texture or letters is somewhat limited and extra draft needs to be incorporated to allow for withdrawal of the part from the mould without damaging the surface.

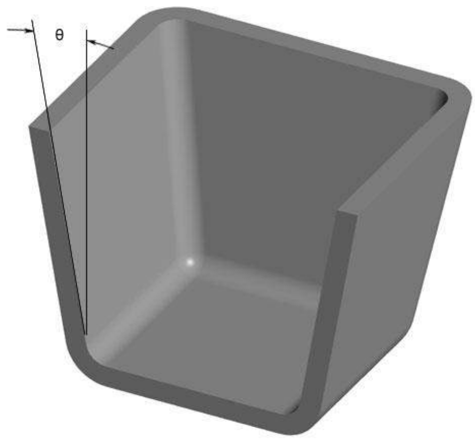
Draft for texturing is somewhat dependant on the mould design and the specific texture requirement. As a general guideline, 1.5 degrees minimum per 0.025mm depth of texture needs to be allowed for in addition to the normal draft. More draft may be needed for heavier textures such as leather grain with a depth of 0.125mm that requires a minimum angle of 7.5 degrees.

There is usually a higher cost associated with aesthetic mirror type finished surfaces as these require greater labour input with regard to the polishing of the mould.

No Draft Angle – Incorrect



Draft Angle – Correct

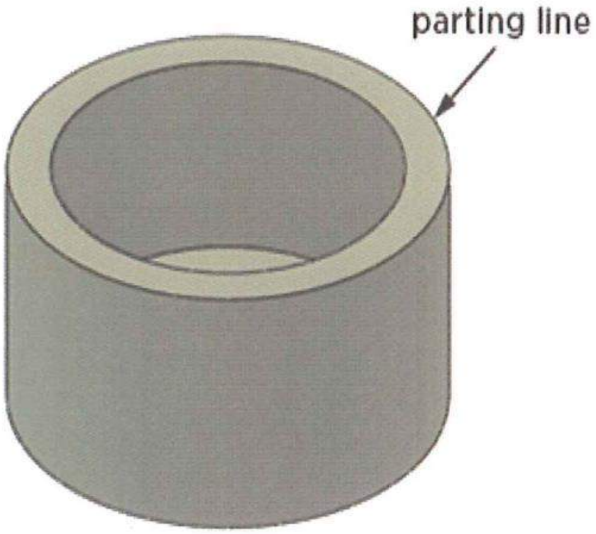


SECTION 2 - PRODUCT DESIGN TIPS

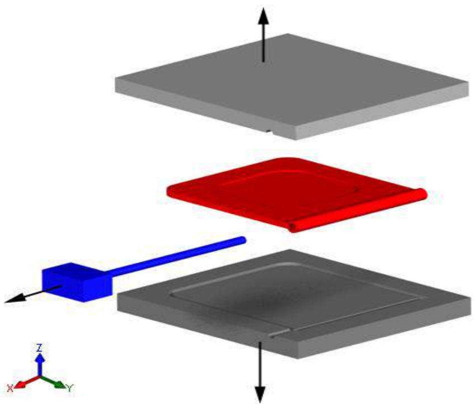
Parting Line

When developing a 3D CAD model to be injection moulded, you may wish to spend some time thinking about where its parting line will be as the location can affect your part in several ways.

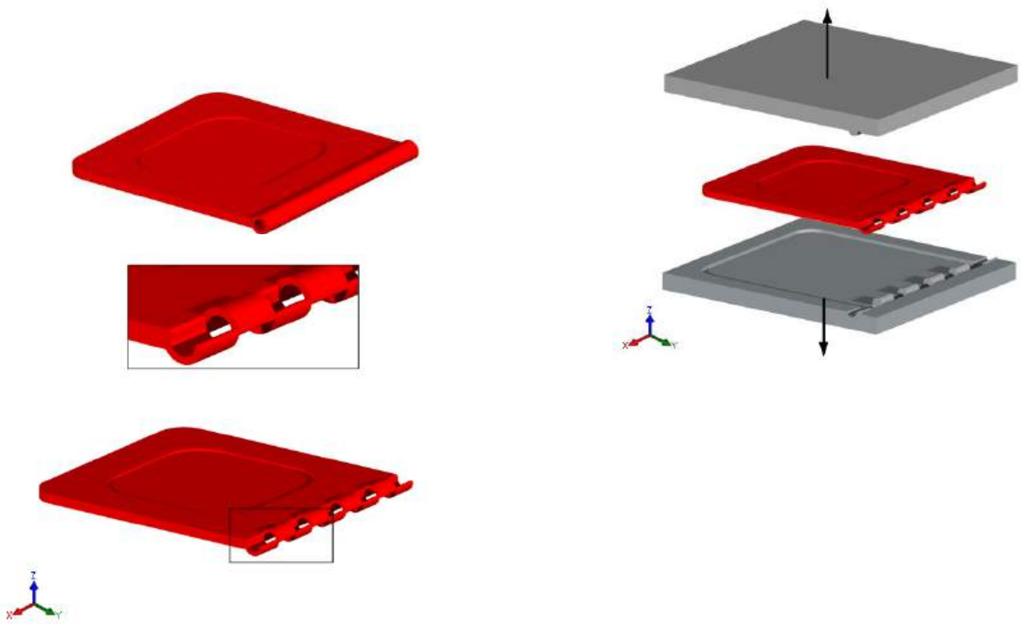
For some parts the location for the parting line is obviously right down the middle, while for more complex parts it may not be so obvious. Take a simple cup for example. The outer face is formed by one mould half while the cup's inner surface and brim will be formed by the other. The parting line occurs along the outside edge of the brim of the cup.



A part that can be made with a straight pull mould has all its features designed so that when the two halves of the mould pull straight away from each other, there is no mould metal that wants to pass through the plastic part. Undercuts on the part require pieces to pull out sideways, perpendicular to the direction of pull (see below) and add significant cost to tooling.

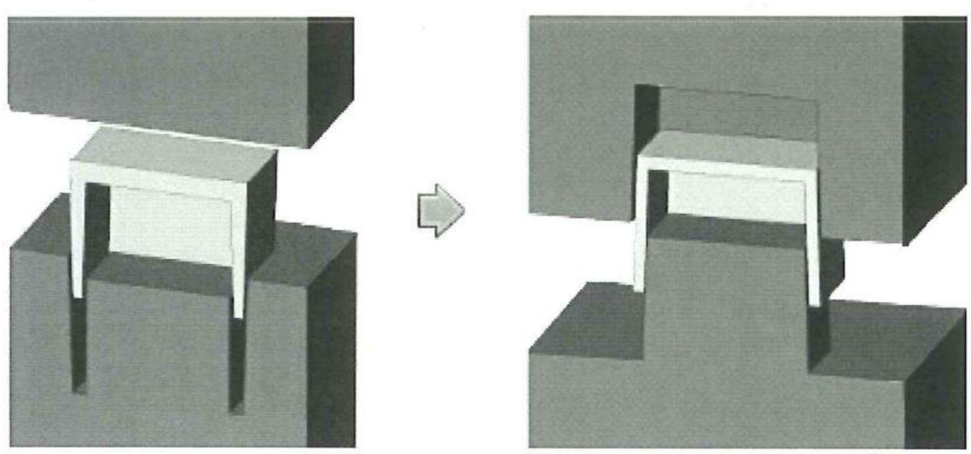


Whilst not always avoidable, the necessity of a side movement (or action) can sometimes be designed out as follows:



Core & Cavity

A mould tool consists of two primary components, the ejection half and the ejector half. Plastic resin enters the mould tool through a sprue in the injection half; the sprue bushing seals tightly against the nozzle of the injection barrel of the moulding machine to allow molten plastic to flow into the cavity. The sprue bushing directs the molten plastic to the cavity through channels that are machined into the ejection and injection halves. These channels are referred to as runners. The molten plastic flows through the runner and enters one or more specialised gates and into the cavity to form the desired part.



Deep Rib Approach
Box designed with walls as ribs. Higher cost to machine and polish cavity.

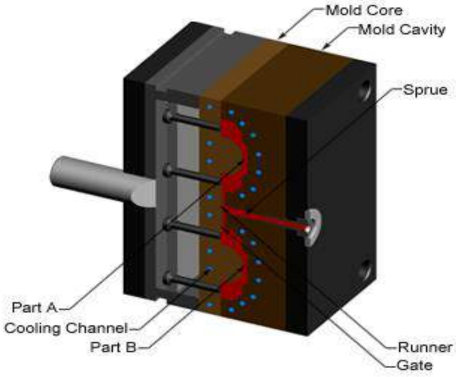
Core Cavity Approach
Box designed thru core-cavity method. Can get to features with bigger faster cutter. Easier and faster to polish.

The Mould Core-Cavity Approach

Consider core-cavity approaches when possible as an alternative to deep sections. The illustrated core-cavity produces a similar geometry but this mould will be faster and less expensive to manufacture. The quality of the final part will be improved to as polishing is much easier on core-cavity geometries than down inside deep sections.

Feed

In order for the molten plastic to flow into the mould cavities, several channels are integrated into the mould tool design. First, the molten plastic enters the mould through a sprue. Additional channels called runners carry the molten plastic from the sprue to all of the cavities that must be filled.



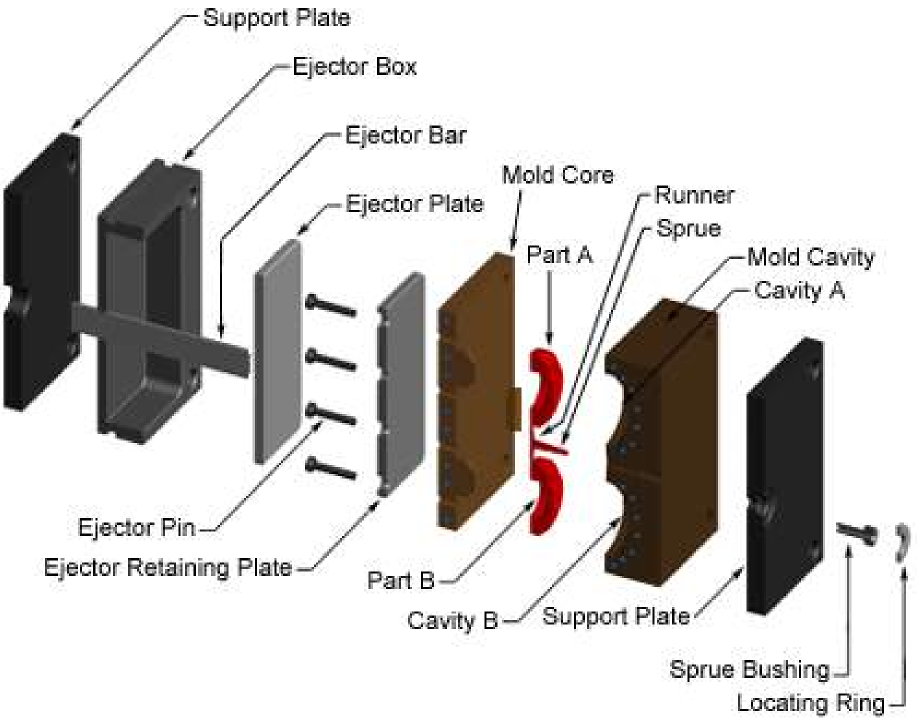
At the end of each runner, the molten plastic enters the cavity through a gate which directs the flow. The molten plastic that solidifies inside these runners is attached to the part and must be separated after the part has been ejected from the mould. However, sometimes hot runner systems are used which independently heat the channels allowing the contained material to remain molten and detached from the part.

Another type of channel that is built into the mould tool is cooling channels. These channels allow water to flow through the mould walls adjacent to the cavity and cool the molten plastic.

Ejection

The injection moulding process mainly uses pins to eject parts from the mould tool during production. Designers should be aware of the need to accommodate such pins. Pins of various sizes are used to push the plastic part out of the mould after it has solidified. The sizes and arrangement of these pins are selected to minimise the impact on the part design.

Sleeves, stripper plates, up/away, up/in and bump off are also alternative methods of ejection.



SECTION 3 - MATERIAL CONSIDERATIONS

There are many types of materials that may be used in the injection moulding process.

Most polymers may be used including all thermoplastics, some thermosets and some elastomers. When these materials are used in the injection moulding process, their raw form is usually small pellets or a fine powder. Also, colourants may be added in the process to control the colour of the final part.

The selection of a material for creating injection moulded parts is not solely based upon the desired characteristics of the final part. While each material has different properties that will affect the strength and function of the final part, these properties also dictate the parameters used in processing these materials. Each material requires a different set of processing parameters in the injection moulding process, including the injection temperature, injection pressure, mould temperature, ejection temperature and cycle time.

A comparison of some commonly used materials is shown below:

Material Name	Abbreviation	Properties	Applications
Acetal	POM	Strong, rigid, excellent fatigue resistance, excellent creep resistance, chemical resistance, moisture resistance, naturally opaque white, low/medium cost	Bearings, cams, gears, handles, plumbing components, rollers, rotors, slide guides, valves
Acrylic	PMMA	Rigid, brittle, scratch resistant, transparent, optical clarity, low/medium cost	Display stands, knobs, lenses, light housings, panels, reflectors, signs, shelves, trays
Acrylonitrile Butadiene Styrene	ABS	Strong, flexible, low mould shrinkage (tight tolerances) chemical resistance, electroplating capability, naturally opaque, low/medium cost	Automotive (consoles, panels, trim, vents) boxes, guages, housings, inhalors, toys
Cellulose Acetate	CA	Tough, transparent, high cost	Handles, eyeglass frames
Polyamide 6 (Nylon)	PA6	High strength, fatigue resistant, chemical resistant, low creep, low friction, almost opaque/white, medium/high cost	Bearings, bushings, gears, rollers, wheels
Polyamide 6/6 (Nylon)	PA6/6	High strength, fatigue resistant, chemical resistant, low creep, low friction, almost opaque/white, medium/high cost	Handles, levers, small housings, zip ties
Polyamide 11+12 (Nylon)	PA11+12	High strength, fatigue resistant, chemical resistance, low creep, low friction, almost opaque to clear, very high cost	Air filters, eyeglass frames, safety masks
Polycarbonate	PC	Very tough, temperature resistance, dimensional stability, transparent, high cost	Automotive (panels, lenses, consoles) bottles, containers, housings, light covers, reflectors, safety helmets and shields
Polyester – Thermoplastic	PBT, PET	Rigid, heat resistance, chemical resistance, medium/high cost	Automotive (filters, handles, pumps) bearings, cams, electrical components (connectors, sensors) gears, housings, rollers, switches, valves
Polyether Sulphone	PES	Tough, very high chemical resistance, clear, very high cost	Valves

Polyetheretherketone	PEEK	Strong, thermal stability, chemical resistance, abrasion resistance, low moisture absorption	Aircraft components, electrical connectors, pump impellers, seals
Polyetherimide	PEI	Heat resistance, flame resistance, transparent (amber colour)	Electrical components (connectors, boards, switches) covers, sheilds, surgical tools
Polyethylene – Low Density	LDPE	Lightweight, tough and flexible, excellent chemical resistance, natural waxy appearance, low cost	Kitchenware, housings, covers and containers
Polyethylene – High Density	HDPE	Tough and stiff, excellent chemical resistance, natural waxy appearance, low cost	Chair seats, housings, covers and containers
Polyphenylene Oxide	PPO	Tough, heat resistance, flame resistance, dimensional stability, low water absorption, electroplating capability, high cost	Automotive (housings, panels) electrical components, housings, plumbing components
Polyphenylene Sulphide	PPS	Very high strength, heat resistance, brown, very high cost	Bearings, covers, fuel system components, guides, switches and shields
Polypropylene	PP	Lightweight, heat resistance, high chemical resistance, scratch resistance, natural waxy appearance, tough and stiff, low cost	Automotive (bumpers, covers, trim) bottles, caps, crates, handles, housings
Polystyrene – General Purpose	GPPS	Brittle, transparent, low cost	Cosmetics packaging, pens
Polystyrene – High impact	HIPS	Impact strength, rigidity, toughness, dimensional stability, naturally translucent, low cost	Electronic housings, food containers, toys
Polyvinyl Chloride - Plasticised	PVC	Tough, flexible, flame resistance, transparent or opaque, low cost	Electrical insulation, housewares, medical tubing, shoe soles, toys
Polyvinyl Chloride - Rigid	UPVC	Tough, flexible, flame resistance, transparent or opaque, low cost	Outdoor applications (drains, fittings, gutters)
Styrene Acrylonitrile	SAN	Stiff, brittle, chemical resistance, heat resistance, hydrolytically stable, transparent, low cost	Housewares, knobs, syringes
Thermoplastic Elastomer/Rubber	TPE/R	Tough, flexible, high cost	Bushings, electrical components, seals, washers