

Injection Moulding

Design and Manufacture

Overview

Injection Moulding (or molding to use the US spelling) along with extrusion, ranks as one of the main processes for producing plastic articles. It is a fast process and is used to produce large numbers of identical items from high precision engineering components to disposable consumer goods.

Typical Materials used:

Acrylonitrile-Butadiene-Styrene (ABS), Nylon (PA), Polycarbonate (PC), Polypropylene (PP) and Polystyrene (GPPS)

Typical Products Produced:

Power-tool housing, Telephone handsets, Television Cabinets, Electrical Switches, DVDs, Automotive bumpers, Automotive dash boards, Battery casings, Syringes, Drug Inhalation Units, Disposable razors, Washing-up bowls, Wheelie Bins, Crates/ Recycling boxes and bottle Lids/ closures

Process:

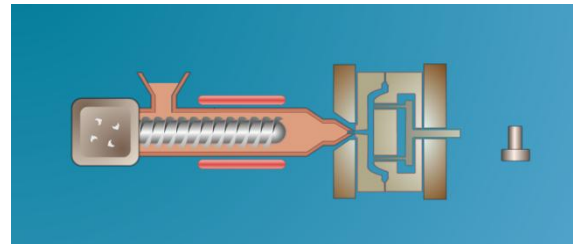
1. Mould fills via gates and runners.

Material is introduced into the injection moulding machine via a Hopper. The injection moulding machine consists of a heated barrel equipped with a reciprocating screw (driven by a hydraulic or electric motor), which feeds the molten polymer into a temperature controlled split mould via a channel system of gates and runners.



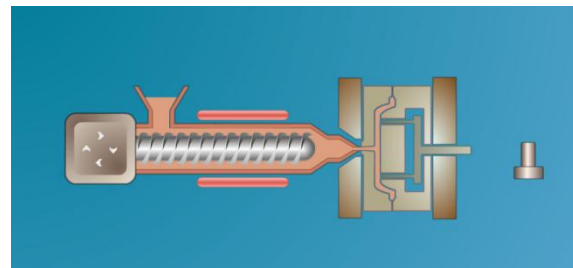
The screw melts (plasticises) the polymer, and also acts as a ram during the injection phase. The screw action also provides additional heating by virtue of the shearing action on the polymer.

The polymer is injected into a mould tool that defines the shape of the moulded part.

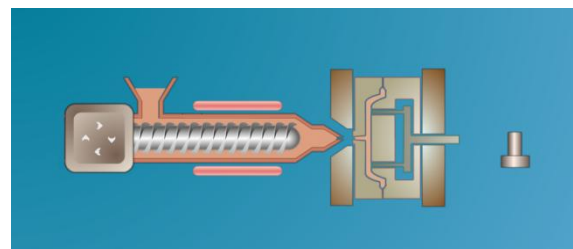


2. Mould packing stage under injection pressure.

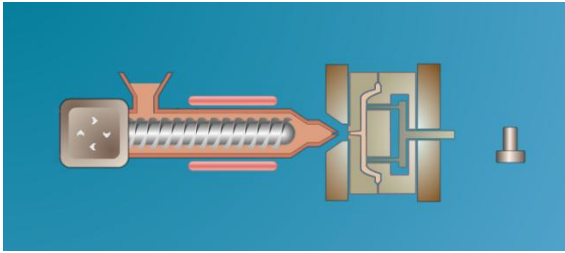
The pressure of injection is high, dependant on the material being processed; it can be up to one thousand atmospheres. Tools tend to be manufactured from steels, (which can be hardened and plated), and Aluminium alloys for increased cutting and hand polishing speeds. The costs associated with tool manufacture means that injection moulding tends to lend itself to high volume manufacture.



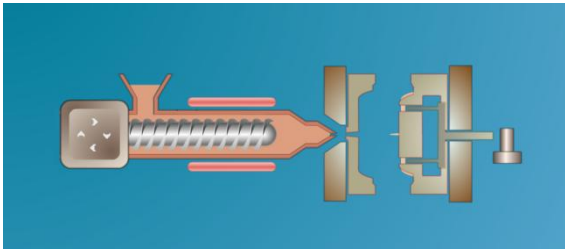
3. Injection unit retracts.



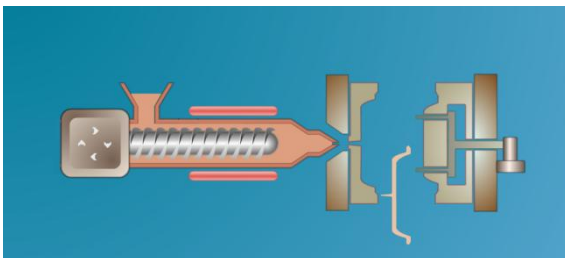
4. Screw recharges barrel with another shot of plastic. Mould coolant cools and solidifies the plastic.



5. Machine opens ready for ejection of component. Ejector bar strikes stop and activates ejector mechanism.



6. The component drops out of mould. Sometimes this is a robotized operation.



The tool can be used to manufacture one consistent part in a repeating process or incorporate multi cavities (a multi impression tool), that is many components can be manufactured on the same tool repeatedly with a single injection.

Variants of the injection moulding process include multi-shot (or 2K moulding) (where different materials are injected into the same mould), insert moulding (where metal components are incorporated), structural foam moulding (where the material is foamed to reduce density) and assisted moulding

(where gas or water are incorporated to reduce wall thickness).

Size of Machine

On most machines the mould is held closed by a combination of toggle clamping and hydraulic pressure. This is known as the locking force. In a large machine this can be up to thousands of tons and is spread over the area of the mould. To counteract the injection pressure, a moulding with a large surface area will therefore need a machine with a higher locking force than a moulding with smaller surface area.

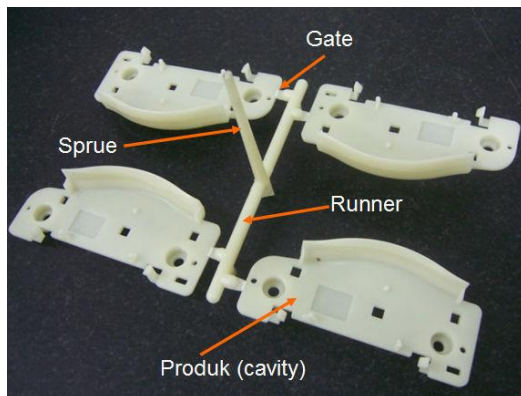
Computer Controlled

Modern day injection moulding machines are controlled by a built-in computer. Acting on sensor fed information, it controls all the actions of the machine and ensures consistent output and shot to shot quality.

Identifying Features

Sprue Marks

In injection moulding, the term sprue refers to the passages through which a liquid material (such as polystyrene or polyvinyl chloride) flows into a die, where it solidifies to form parts. It also refers to the material which solidifies in these passages, forming a framework attaching the parts together in a roughly planar arrangement.

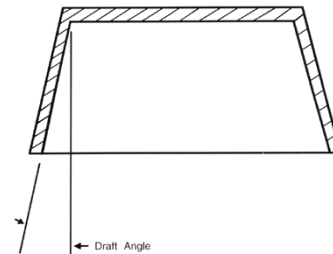


Some mould makers make a distinction between three separate entities: the gate, the runner, and the sprue. Certainly in the plastics injection moulding industry, the gate is the location at which the molten plastic enters the mould cavity and is often seen as a small nub or projection (the "gate mark") on the moulded piece. The sprues are large-diameter channels through which plastic flows, usually around the edges of the part or along straight lines. Finally, in this naming scheme, the runner represents only the smaller channels that divert from the sprue to the individual part. An analogy to the sprue/runner system might represent the sprues with city water mains, and runners with the smaller pipes leading to individual houses.

Many scale model kits are made from injection-moulded plastic.

Draft Angles

Plastic parts are always designed with a taper or draft in the direction of mould movement to allow part ejection or removal from the mould. Since plastics shrink when cooled, it is common for parts to shrink (or grip tightly) to cores. A good definition for draft would be: the degree of taper of a side wall or rib needed to allow the moulded plastic part to be removed from the metal mould. Without proper draft, plastic parts may be difficult to remove from the mould



Mould Split Lines

A *parting line* in mould making is the place where two or more parts of the mould meet. Parting lines occur due to minute inherent gaps between two mating faces of halves of the die. The material filling into these gaps is generally called flashing. Parting lines generally show up as a raised line around the product on commodity products such as toys.

Many applications (seals, tight running moulded parts) that require precision for shape control, call for removal of flashes. Secondary operations are thus employed, in addition to high precision moulding dies to remove these flashes.

Ejection pin Marks

Ejector mark is a concave or a convex on the surface of plastics product. It causes rapid variation in thickness which leads to gloss differences and concaves.



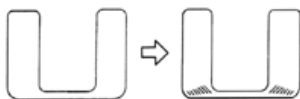
Uniform wall thickness

If there was only one rule for the injection moulding process it would have to be to maintain uniform wall thickness. Here are some examples of problems associated with part designs that feature a non-uniform wall thickness.

Sink marks due to uneven shrinkage - Sink marks result from a wall yielding to the still shrinking interior mass.



Stress due to uneven shrinkage - A part with non-uniform wall thickness will cool unevenly, resulting in high moulded-in stress.



Voids due to uneven shrinkage - The already cooled section will not yield to the shrinking action of the cooling interior mass causing voids in the thick portion of the part.



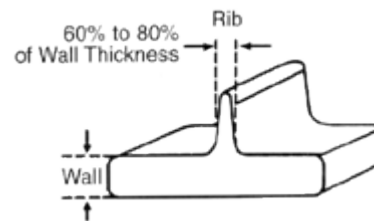
Warping due to uneven shrinkage -

Concentrated stress at the junction of high & low shrinkage area may cause a part to warp.



Webs/Ribs for Strength

Many times the stiffness of a part must increase because of the load applied to the part design. One of the easiest ways to cure this problem is change the part geometry by adding ribs. The use of ribs is a practical way and economical means of increasing the structural strength of a part. But there are guidelines that govern adding ribs without causing sink marks or surface blemishes to your parts.



Bosses

Bosses are used for locating, mounting, and assembly purposes. There are boss design guidelines that must be followed to insure the highest quality in moulded parts. Again, one of the main points to consider is nominal wall thickness. Too many times bosses are designed with thick wall sections that can affect the appearance of the plastic part and the final product.

