INJECTION MOLDING

SUBMITTED BY:

MANEET KOHLI
ROLL NO. 11

INJECTION MOLDING

Injection molding is nowadays the most popular method to produce 3-dimensional parts of different kinds of polymeric materials. It is a fast process and is used to produce large numbers of identical items from high precision engineering components to disposable consumer goods. It is a suitable method for thermoplastics, thermosets, thermoplastic elastomers, elastomers, short fiber and particulate filled polymers. Today, more than one-third of all thermoplastic materials are injection molded and more than half of the polymer processing equipment is for injection molding. Injection molding is a production method for large series and it can be characterized as follows:

- part shape can be very complex and it has no specific limitations
- part size is not limited (from very small parts to quite big products)
- high fundamental investigations
  - injection molding machines (300 000 FMk and more)
  - injection moulds (50 000 FMk and more)
- short cycle time
- very good dimensional stability
- very good visual quality
- integration of different post processing methods (coating, inserts) possible

Typical Material Used

Most thermoplastics can be processed by Injection Molding. Some of the commonly used materials are:

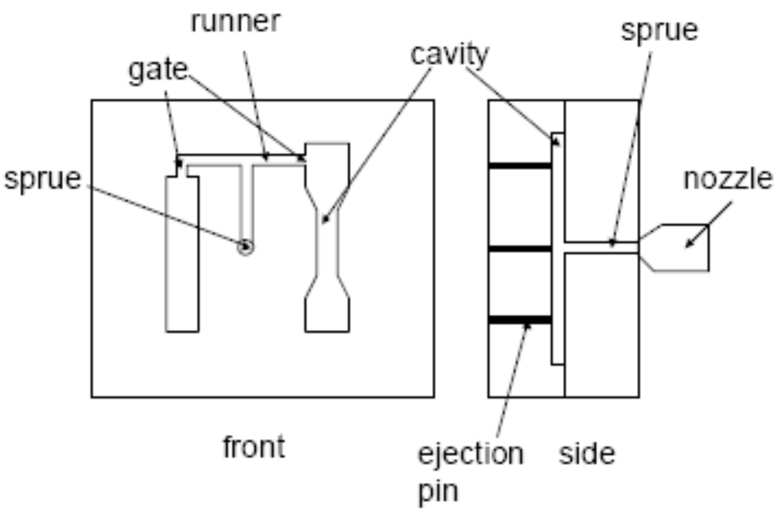
- Acrylonitrile-Butadiene-Styrene
- Nylon
- Polycarbonate
- Polypropylene
- Polystyrene

Injection molding can also be used to manufacture parts from aluminium or brass. The melting points of these metals are much higher than those of plastics; this makes for substantially shorter mold lifetimes despite the use of specialized steels. Nonetheless, the costs compare quite favorably to sand casting, particularly for smaller parts.

Equipment

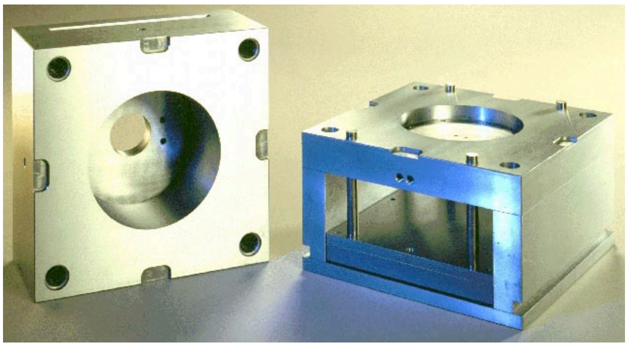
Injection molding machines, also known as presses, hold the molds in which the components are shaped. Presses are rated by tonnage, which expresses the amount of clamping force that the machine can generate. This pressure keeps the mold closed during the injection process. Tonnage can vary from less than 5 tons to 6000 tons, with the higher figures used in comparatively few manufacturing operations.

MOLD



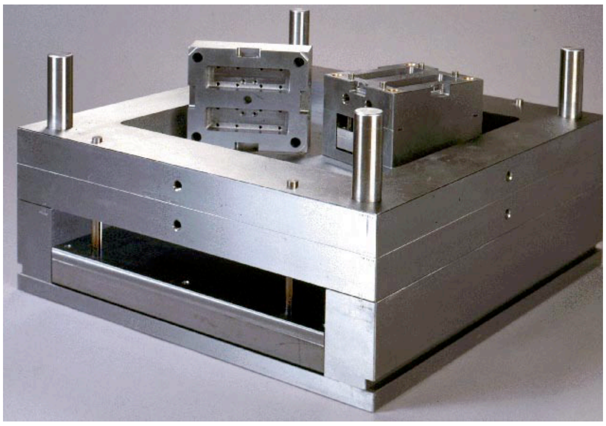
Mold (Tool and/or Mold) is the common term used to describe the production tooling used to produce plastic parts in molding.

Traditionally, molds have been expensive to manufacture. They were usually only used in mass production where thousands of parts were being produced. Molds are typically constructed from hardened steel, pre-hardened steel, aluminium, and/or beryllium-copper alloy. The choice of material to build a mold is primarily one of economics. Steel molds generally cost more to construct, but their longer lifespan will offset the higher initial cost over a higher number of parts made



before wearing out.

Pre-hardened steel molds are less wear resistant and are used for lower volume requirements or larger components. The steel hardness is typically 38-45 on the Rockwell-C scale. Hardened steel molds are heat treated after machining. These are by far the superior in terms of wear resistance and lifespan. Typical hardness ranges between 50 and 60 Rockwell-C (HRC). Aluminium molds can cost substantially less, and when designed and machined with modern computerized equipment, can be economical for molding tens or even hundreds of thousands of parts. Beryllium copper is used in areas of the mold which require fast heat removal or areas that see the most shear heat generated.



DESIGN

Molds separate into two sides at a parting line, the *A side*, and the *B side*, to permit the part to be extracted. Plastic resin enters the mold through a *sprue* in the A plate, branches out between the two sides through channels called *runners*, and enters each part *cavity* through one or more specialized *gates*. Inside each cavity, the resin flows around protrusions (called *cores*) and conforms to the cavity geometry to form the desired part. The amount of resin required to fill the sprue, runner and cavities of a mold is a *shot*. When a core *shuts off* against an opposing mold cavity or core, a hole results in the part. Air in the cavities when the mold closes escapes through very slight gaps between the plates and pins, into shallow plenums called *vents*. To permit removal of the part, its features must not overhang one another in the direction that the mold opens, unless parts of the mold are designed to move from between such overhangs when the mold opens. Sides of the part that appear parallel with the direction of *draw* (the direction in which the core and cavity separate from each other) are typically angled slightly with (*draft*) to ease release of the part from the mold, and examination of most plastic household objects will reveal this. Parts with bucket-like features tend to shrink onto the cores that form them while cooling, and cling to those cores when the cavity is pulled away. The mold is usually designed so that the molded part reliably remains on the ejector (B) side of the mold when it opens, and draws the runner and the sprue out of the (A) side along with the parts. The part then falls freely when ejected from the (B) side. *Tunnel gates* tunnel sharply below the parting surface of the B side at the tip of each runner so that the gate is sheared off of the part when both are ejected.

Ejector pins are the most popular method for removing the part from the B side core(s), but air ejection, and stripper plates can also be used depending on the application. Most ejector plates are found on the moving half of the tool, but they can be placed on the fixed half if spring loaded. For thermoplastics, coolant, usually water with corrosion inhibitors, circulates through passageways bored through the main plates on both sides of the mold to enable temperature control and rapid part solidification.

To ease maintenance and venting, cavities and cores are divided into pieces, called *inserts*, and subassemblies, also called *inserts*, *blocks*, or *chase blocks*. By substituting interchangeable inserts, one mold may make several variations of the same part.

More complex parts are formed using more complex molds. These may have sections called *slides* that move into a cavity perpendicular to the draw direction, to form overhanging part features. Slides are then withdrawn to allow the part to be released when the mold opens. Slides are typically guided and retained between rails called *gibs*, and are moved when the mold opens and closes by angled rods called *horn pins* and locked in place by *locking blocks*, both of which move cross the mold from the opposite side.

Some molds allow previously molded parts to be reinserted to allow a new plastic layer to form around the first part. This is often referred to as over molding. This system can allow for production of one-piece tires and wheels.

2-shot or multi shot molds are designed to "over mold" within a single molding cycle and must be processed on specialized injection molding machines with two or more injection units. This can be achieved by having pairs of identical cores and pairs of different cavities within the mold. After injection of the first material, the component is rotated on the core from the one cavity to another. The second cavity differs from the first in that the detail for the second material is included. The second material is then injected into the additional cavity detail before the completed part is ejected from the mold. Common applications include "soft-grip" toothbrushes and freelander grab handles.

The core and cavity, along with injection and cooling hoses form the mold tool. While large tools are very heavy (up to 60t), they can be hoisted into molding machines for production and removed when molding is complete or the tool needs repairing.

A mold can produce several copies of the same parts in a single "shot". The number of "impressions" in the mold of that part is referred to as cavitation. A tool with one impression will often be called a single cavity (impression) tool. A mold with 2 or more cavities of the same parts will likely be referred to as multiple cavity tooling. Some extremely high production volume molds (like those for bottle caps) can have over 128 cavities.

In some cases multiple cavity tooling will mold a series of different parts in the same tool. Some toolmakers call these molds family molds as all the parts are not the same but often part of a family of parts (to be used in the same product for example).

MACHINING

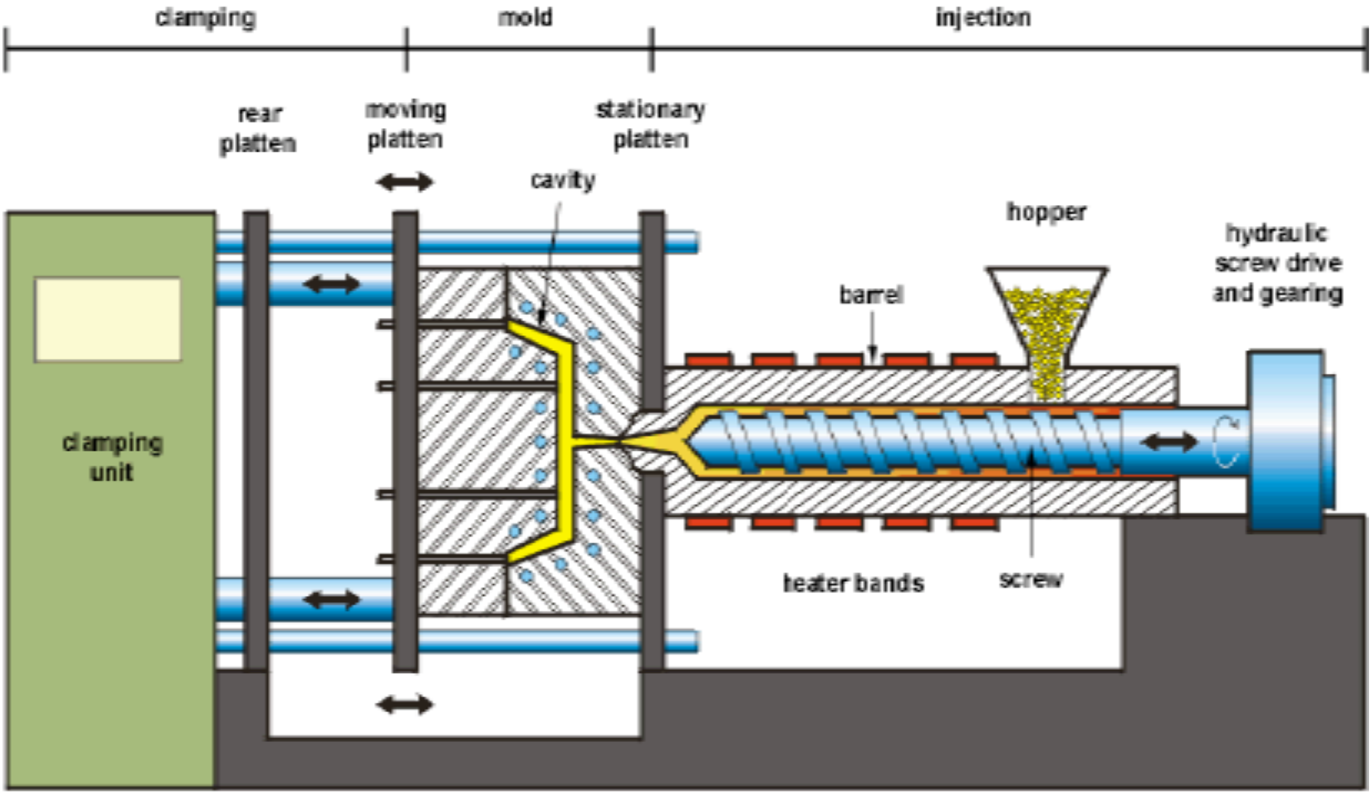
Molds are built through two main methods: **standard machining** and **EDM machining**. Standard Machining, in its conventional form, has historically been the method of building injection molds. With technological development, CNC machining became the predominant means of making more complex molds with more accurate mold details in less time than traditional methods.

The electrical discharge machining (EDM) or spark erosion process has become widely used in mold making. As well as allowing the formation of shapes which are difficult to machine, the process allows pre-hardened molds to be shaped so that no heat treatment is required. Changes to a hardened mold by conventional drilling and milling normally require annealing to soften the steel, followed by heat treatment to harden it again. EDM is a simple process in which a shaped electrode, usually made of copper or graphite, is very slowly lowered onto the mold surface (over a period of many hours), which is immersed in paraffin oil. A voltage applied between tool and mold causes erosion of the mold surface in the inverse shape of the electrode.

COST

The cost of manufacturing molds depends on a very large set of factors ranging from number of cavities, size of the parts (and therefore the mold), complexity of the pieces, expected tool longevity, surface finishes and many others.

THE PROCESS



1. Clamping

An injection molding machine consists of three basic parts; the mold plus the clamping and injection units. The clamping unit is what holds the mold under pressure during the injection and cooling. Basically, it holds the two halves of the injection mold together.

2. Injection

During the injection phase, plastic material, usually in the form of pellets, are loaded into a hopper on top of the injection unit. The pellets feed into the cylinder where they are heated until they reach molten form (think of how a hot glue gun works here). Within the heating cylinder there is a motorized screw that mixes the molten pellets and forces them to end of the cylinder. Once enough material has accumulated in front of the screw, the injection process begins. The molten plastic is inserted into the mold through a sprue, while the pressure and speed are controlled by the screw.

Note: Some injection molding machines use a ram instead of a screw.

3. Dwelling

The dwelling phase consists of a pause in the injection process. The molten plastic has been injected into the mold and the pressure is applied to make sure all of the mold cavities are filled.

4. Cooling

The plastic is allowed to cool to its solid form within the mold.

5. Mold Opening

The clamping unit is opened, which separates the two halves of the mold.

6. Ejection

An ejecting rod and plate eject the finished piece from the mold. The un-used sprues and runners can be recycled for use again in future molds.

The injection molding process requires some complex calculations. Every different type of resin has a shrinkage value that must be factored in, and the mold must compensate for it. If this value is not precisely determined, the final product will be incorrectly sized or may contain flaws. Typically, this is compensated for by first filling the mold with resin, holding it under pressure, and then adding more resin to compensate for contraction. Other complications may include burned parts resulting from the melt temperature being set too high, warpage resulting from an uneven surface temperature, or incomplete filling due to a too slow of an injection stroke.

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

Injection molding is a complex technology with possible production problems. They can either be caused by defects in the molds or more often by part processing (molding)

Molding Defects	Alternative name	Descriptions	Causes
Blister	Blistering	Raised or layered zone on surface of the part	Tool or material is too hot, often caused by a lack of cooling around the tool or a faulty heater
Burn Marks	Air Burn/ Gas Burn	Black or brown burnt areas on the part located at furthest points from gate	Tool lacks venting, injection speed is too high
Color Streaks		Localized change of color	Master batch isn't mixing properly, or the material has run out and it's starting to come through as natural only
Delamination		Thin mica like layers formed in part wall	Contamination of the material e.g. PP mixed with ABS, very dangerous if the part is being used for a safety critical application as the material has very little strength when delaminated as the materials cannot bond
Flash	Burrs	Excess material in thin layer exceeding normal part geometry	Tool damage, too much injection speed/material injected, clamping force too low
Embedded contaminates	Embedded Particulates	Foreign particle (burnt material or other) embedded in the part	Particles on the tool surface, contaminated material or foreign debris in the barrel, or too much shear heat burning the material prior to injection
Flow marks		Directionally "off tone" wavy lines or patterns	Injection speeds too slow (the plastic has cooled down too much during injection, injection speeds must be set as fast as you can get away with at all times)
Jetting		Deformed part by turbulent flow of material	Poor tool design, gate position or runner. Injection speed set too high.

Polymer degradation		polymer breakdown from hydrolysis, oxidation etc	Excess water in the granules, excessive temperatures in barrel
Silver streaks		Circular pattern around gate caused by hot gas	Moisture in the material, usually when hygroscopic resins are dried improperly
Sink Marks		Localized depression (In thicker zones)	Holding time/pressure too low, cooling time too low, with sprueless hot runners this can also be caused by the gate temperature being set too high
Short shot	Non-Fill / Short mold	Partial part	Lack of material, injection speed or pressure too low
Splay Marks	Splash mark / Silver Streaks	Circular pattern around gate caused by hot gas	Caused by the material (plastic) being damped prior to injection
Stringiness	Stringing	String like remain from previous shot transfer in new shot	Nozzle temperature too high. Gate hasn't frozen off
Voids		Empty space within part (Air pocket)	Lack of holding pressure (holding pressure is used to pack out the part during the holding time). Also mold may be out of registration (when the two halves don't center properly and part walls are not the same thickness).
Weld line	Knit Line / Meld Line	Discolored line where two flow fronts meet	Mold/material temperatures set too low (the material is cold when they meet, so they don't bond)
Warping	Twisting	Distorted part	Cooling is too short, material is too hot, lack of cooling around the tool, incorrect water temperatures (the parts bow inwards towards the cool side of the tool)

Advantages of Injection Molding

- High production rates
- High tolerances are repeatable
- Wide range of materials can be used
- Low labor costs
- Minimal scrap losses
- Little need to finish parts after molding

Disadvantages of Injection Molding

- Expensive equipment investment
- Running costs may be high
- Parts must be designed with molding consideration

Related injection molding processes

During the recent years the injection mounding process has developed a great deal. There are several special injection mounding techniques e.g.

- multi-color
- multi-component
- co-injection
- gas-assisted
- injection-compression
- in-mould decoration
- external inserts

Computer simulation in injection molding

The injection moulding process can be simulated by computer. These programs can calculate

- filling of mould
- melt temperature
- melt pressure
- cooling
- internal stresses
- warpage