Injection Moulding Process – An Overview

Mayuresh Nitin Madiwale

Graduate Mechanical Engineer, Intern at Amol Enterprises, MIDC Satpur Nashik

ABSTRACT

Injection Moulding process is one of the most commonly used among all plastic processing methods due to its advantages over other processes. Injection Moulding Process uses high temperature to melt the charge and pressure to inject molten charge in mould cavity. This paper covers a brief overview of Injection Moulding Process along with process parameters and their influence; mould design parameters, defects seen in moulded part with their remedy and latest trends in Injection Moulding Process

Keywords: - Injection Moulding Process, Thermoplastics, Defects and Remedies, Process Parameters, latest trends.

1. INTRODUCTION

As the demand of plastic products is increasing in the market, the need of supplying more in less time is also increasing in order to maintain the demand and supply chain. This creates an obligation for manufacturers to reduce cycle time and to improve quality. Therefore the conditions are approving for Injection Moulding as its production rate is high, repeatability is high, low production cost, moderate scrap, vast availability of materials as per need and has ability to produce required surface finish and engraved surfaces.

Manufacturers tend towards "one time setting" of all process parameters and try to achieve their daily production schedule so as to maintain and improve their competency in the market which is also possible in this process.

Waste in production process is also an important factor that leads to loss in efficiency of manufacturing unit and financial balance of company. To counter that, the wastage produced in Injection Moulding Process (flash, runners, gates, defective parts) is recycled (Mostly thermoplastic materials).

The pressure in IM is relatively less than compression moulding (CM) process as it is only useful for injection stage thereafter full pressure is not required. Thermoplastics are most commonly used material as this group possesses higher MFI (Melt Flow Index) than that of Thermosets. Thus Thermosets are not suggested to be moulded by this process and another reason being their inability to re-melt. The process starts with granulated raw material which is dried in a drier to remove moisture content and to soften the material before moulding. Then the dried raw material is poured in the hopper of moulding machine from where it moves to the barrel to be melted. After melting, it is then injected into the mould from nozzle to sprue bush to runner of the part and then finally in the cavity where it solidifies and takes the shape of cavity. The barrel is maintained at high temperature to melt the plastic while the mould is maintained at lower temperature to initiate and catalyze solidification by means of running cold water through mould or by simple air circulation around the mould. The ejection system arranged in the mould facilitates the ejection safely. Thus the process completes with ejection of moulded part.

1.1 Requirements of Injection Moulding Process

Injection Moulding process requires following:-

1) Raw material

Plastic or Polymer precisely has three main subtypes namely Thermoplastics, Thermosets and Elastomers. Thermoplastics are recyclable upto certain limit. They can be melted by heating and solidified by cooling. Polypropylene (PP), Polyethylene (PE), Polystyrene (PS), Vinyl, Acrylic, ABS, Nylon, and Polycarbonate (PC) are some TP's used in various applications. On the other hand, Thermosets are not recyclable. They can be heated to melt only for first time after which they cannot be used again. If heated again, they will degrade and will be of no use for moulding thereafter. Phenolic, Polyester, Melamine, Urethane, Epoxy, Alkyd, Diallyl are some TS's available for various applications. Elastomers group contains polymers with high elasticity. These are also engineering materials used in various applications which require different processes. Rubber, Butyl, Silicone, Fluorocarbon, Polysulfide, Polysulfide, Neoprene, etc are some Elastomers used in day to day application.

Thus, Thermoplastics are widely used in Injection Moulding process while Thermosets are moulded by Compression Moulding process.

Raw Thermoplastic is available in granules, powders, flakes, tapes, pellets or liquids. These are mainly available in clear or white base colour. To add colour, master batches are added in specific amounts. Master batches can be pigments or dyes. Raw material is dried in a drier to eliminate the moisture content in plastic and to soften them before melting.

Fig -1 Raw Material Granules

2) Injection Mould

An accurate and precise mould cavity makes a perfect product. Mould has three main components, Cavity, Core (Punch) and Ejection system. Cavity is the pocket in mould where molten plastic gets filled and solidifies. Punch covers the cavity and or makes desired alignment with cavity to produce a hole in product. Ejector system helps eject the final product from mould. Cavity is on one side of mould hence it is called cavity half similarly, punch is on one side of mould, so it is called punch half. Ejector system is usually arranged behind cavity side.

Mould material plays a vital role in accuracy of product. Components of mould that come in direct contact of plastic such as Punch plate, Cavity plate, Stripper plate are made to withstand heat, pressure and wear-tear, thus they are made with strong metal alloys such as HDS (Hot Die Steel) or OHNS. Guide pins and ejector pins are also made with same material. Other parts such as Top plate, Bottom plate, Punch back plate, Cavity back plate, Ejector plates, and spacers are made with softer material like MS (Mild Steel).

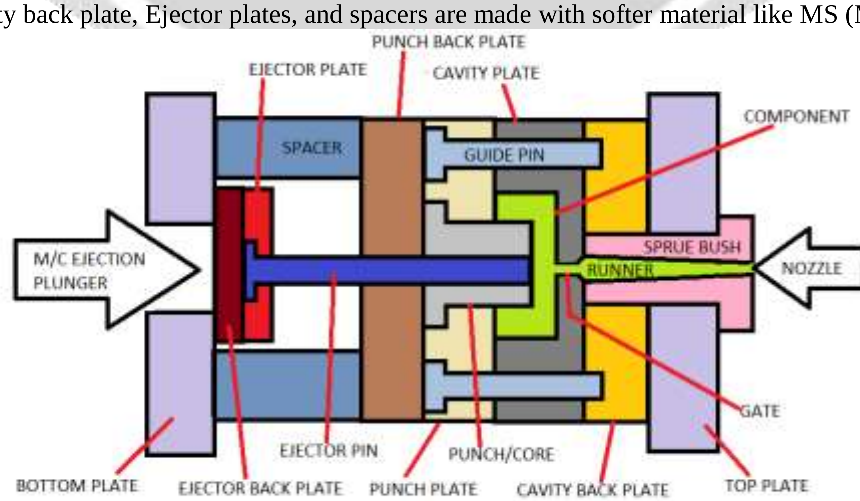


Fig -2 Schematic of Injection Mould

3) Injection Moulding machine

Injection moulding machines are available in two types, Vertical Moulding machines, and Horizontal Moulding machines. This is distinguished by the direction of molten plastic through which it enters the mould. Vertical machines usually require less floor space and are used for small light weighted products while the Horizontal machines require more floor space and are used for heavy and large products. Modern moulding machines (both vertical and horizontal) usually are fully automatic which theoretically do not require any attention from workers.
But in practical applications, they are operated as semi automatic machines requiring some attention. E.g.:- removing product and runners, adjusting various pressures as per need, lubricating the mould if the product is sticking to it, etc. In Vertical Moulding machines plastic enters the mould through a channel carved between punch and cavity plates. This method is useful only in Vertical Moulding machines; it does not require sprue or any auxiliary components. In Horizontal Moulding machines, plastic enters through punch side through a channel or hole drilled. This requires sprue, locating ring and proper channeling to the cavity.

Fig -3 Horizontal Injection Moulding Machine

2 PROCESS FLOW

1) Raw material procurement

Raw material in form of granules, sheets, tapes are purchased as per requirement and inspected upon arrival. Incoming inspection is done to assure that the material bags aren't tampered with or damaged in any way. Also the grade and quantity are checked to be sure. Finally it is stored in its appropriate location.

2) Pre-Moulding process

a) Storage

Purchased raw material needs to be stored in a well-ventilated dry place away from direct sunlight and any heat source. Moisture has a bad impact on moulding process thus it should be away from moisture. Thus the bags should not be opened until it is needed.

b) Colour mixing

Raw material can be procured in two ways, 1) pre-colored with desired colour code or, 2) clear plastic and its separate master batch (Colorant). The choice depends on manufacturer or the client. If master batch is to be added to the clear plastic, it is defined by **Let-Down Ratio (LDR)** which is the ratio of master batch to the base material.
E.g.:- If LDR is 3%, master batch is 3 grams per 100 grams of base material.

1. Color code system:-
   The colorant is defined by many color matching systems one of which is **RAL** value .E.g.:- RAL 7032 – Pebble grey, RAL 1004 – Golden yellow
   RAL is a European color matching system which defines colors for paint, coatings and plastics. The RAL color standard is administrated by the RAL Deutsches Institut für Gütesicherung und Kennzeichnung. 'RAL' is the abbreviation of **'Reichs-Ausschuß für Lieferbedingungen und**

Gütesicherung'. This name can be translated in English as **'National Commission for Delivery Terms and Quality Assurance'.** [1]

c) Drying

After mixing of colorant pigment or granules, the batch is dried in a Blow Drier. This process removes the moisture content from the plastic. This can also be done in a low temperature furnace.

Fig -4 Hopper Drier [4]

3) Moulding process

a) Shot size

In each cycle, a specific amount of plastic is injected into the mould. This amount depends upon the weight of part to be moulded and runner system used in moulding. Extra material in small quantities is always helpful in ensuring the complete filling of cavity.
E.g.:- a part is weighing 50 grams, and runner system is weighing 2 grams, then the shot size is 10+2 =12 grams

b) Homogenization

Plastic granules from hopper are fed into the barrel of injection moulding. The screw in the barrel mixes the plastic thoroughly, this ensures proper color as per requirement and the color is homogenized for the entire batch. After one cycle is completed, the barrel screw turns while moving backwards filling the shot and homogenizing the color simultaneously in this process.

c) Melting

Heaters attached at various points on the barrel melts the plastic. Ceramic, MICA band, induction coil heaters are some types of heaters used in injection moulding process. Modern moulding machines have 5 point heaters, four on the barrel and one on the nozzle.

d) Pressurizing

The screw is then fed linearly pressurizing the charge in the barrel. As the screw moves towards the nozzle, the pressure increases because the nozzle area is far less than the barrel cross-sectional area. The pressure can be controlled with changing the pressure of hydraulic system.

e) Injecting

Pressurized charge moves with high velocity as it passes the nozzle and enters the sprue bush. The internal aperture of sprue bush is tapered to facilitate the easy removal of runner after the solidification of part. This also helps in reducing the jet velocity and increasing pressure similar to a venturi.

f) Solidification

After complete filling of cavity, the plastic starts to solidify as the mould temperature is significantly lower than the barrel. Also the mould temperature can be controlled by using external aid such as water cooling system.

g) Ejection

After complete solidification, the part is ejected or "De-moulded" from the mould. There can be in-built ejection system which is most widely used in practice or there can be external ejection system such as "Pneumatic Ejection" which requires compressed air supply.

Various types of ejection systems include:-

I. Pin ejection
II. Sleeve ejection
III. Stripper plate ejection
   a. Ejection pin operated
   b. Side ejection
IV. Blade ejection
V. Pneumatic ejection (Air ejection)

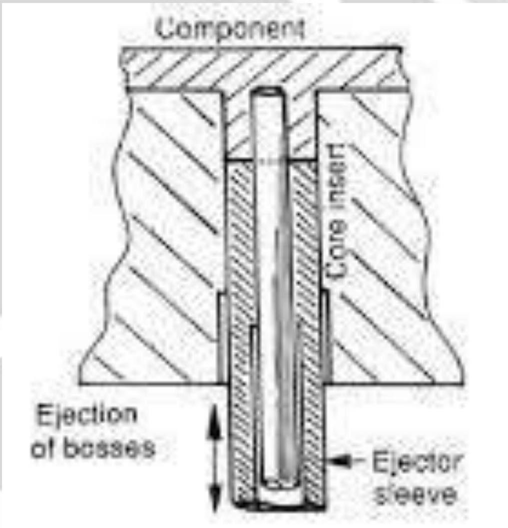


Fig -5 Pneumatic Ejection System Schematic

4) Post-Moulding process

a) De-flashing

Flash/ Burr are difficult to eliminate. Thus the parts are deflashed. Operator removes the flash present on the edges of the part and also removes the runner attached to the parts. This can be done when next cycle is underway so as to eliminate further processing time.

b) Conditioning

Post Moulding Conditioning is a process required by some polymer types like Polyamide (Nylon), Acetal and composites like Glass filled Nylon. These polymers absorb water from atmosphere. Water is a plasticizer of these polymers and is necessary for dimensional and structural stability. The time it takes to absorb moisture from atmosphere and attain equilibrium can be few days, weeks or even months. Thus conditioning is necessary as soon as the part is ejected. Standard procedure suggests the conditioning in water at 80-85º C for at least an hour.

c) Storage

After above steps are carried out, parts are dried again if there is moisture present and stored in a well ventilated, dry room before it can be shipped to the customer.

3. PROCESS PARAMETERS

There are various process parameters that control the operations. Also Mould design plays a significant role in quality of part moulded. Following are some process parameters:-

I. Melt temperature
It is the temperature of the melted plastic. If the temperature is low, the plastic will not evenly melt and may get stuck in the nozzle or gate area. If too high, it will degrade during the cycle and will evaporate the moisture trapped and will be prone to trapped air bubbles.

II. Injection pressure
It is the pressure applied by plunger on the charge. If too high, it will show flow marks in the part and if too low, the mould cavity will not fill completely.

III. Ejection pressure
It is the pressure applied by the ejector rod on the ejection system and on the part ultimately. If too low, part will take longer to leave the cavity. If too high, the part will get deformed and may get scored by the walls of cavity.

IV. Back pressure
It is the pressure applied on the screw on the way to collect new material from hopper. For proper homogenization, this pressure should be controlled properly.

V. Screw RPM
It is the speed in revolutions per minute for the screw while it mixes the material for homogenization. If the speed is low, the material may not mix properly causing colour defects. If too high, it will mix unnecessary air in the material causing blow holes.

VI. Charge mass
It is the quantity of material imported in the barrel from hopper. This quantity should be equal to the mass of part, runner system and little bit extra to ensure complete filling of cavity. More quantity will affect the part as it will create flash. Less material will lead to short shot.

VII. Hold time / cooling time
It is the time for which the mould is kept closed after injection is complete. During this time, the part solidifies inside the cavity. If this time is less, the part will not solidify completely and will get deformed on removal. If it is more than required, it will increase cycle time.

VIII. Fill time
It is the time for which the injection is carried out. If enough time is not provided, it will cause incomplete filling of cavity. If more, it will cause more material to get inside cavity causing excessive flash.

IX. Injection speed
It is the jet velocity of molten plastic. If it is more, it will cause flow pattern on part, which is a problem when part is transparent. To avoid this runner design can be made such that it will not let high velocity jet to go inside cavity.

X. Ejection speed
It is the speed by which the ejection system will operate. If the speed is more than requirement, it will cause deformation on part. If the speed is less it will increase the overall cycle time of moulding.

XI. Clamping force
It is the force applied by the machine on the mould halves for proper sealing of mould. If less, it may leave gap between the mould plates where material will get filled and flashes would occur. If too high, it will cause damage to the hydraulics of machine and the mould surfaces.

4. MOULD DESIGN PARAMETERS

These are the parameters to be considered while designing the mould. These parameters affect the part quality as well.

I. Number of gates
Number of gates in a product influences its mechanical properties as well as the cycle time. If more than one gate is present in the part, there will be weld line created compulsorily. There can be no such part moulded with a number of gates and no weld line. And when weld line gets created, the structural rigidity will be compromised specifically when the part is to be used where loads will be applied. To reduce this, the gates should be placed such that the weld line will be on non critical areas of the part. However the cycle time will be reduced dramatically when there is more than one gate.

II. Gate size
If there is obligation that number of gates cannot be made and the cycle time should be less, then increasing the gate size can to the trick. More cross sectional area of gate means fast filling of part but much larger portion of part needs to be trimmed of when runner system is being cut. Thus size of gate should be appropriate such that final part does not look shabby on gate location.

III. Part's geometric complexity
Geometric complexity of part matters a lot when moulding is being done. A very complex part will take more time in mould design and manufacturing and hence will cost more. So it should be clear that the part when designed for a specific condition should be less complex and easy to mould.

IV. Mould material
For a long run mould material should be capable of handling loads, abrasion and wear and tear. A very simple mould which will not be used for mass production can be made from mild steel which is cheap as well as capable for handling load for short duration. On the other hand, most moulds are made with harder materials such as P20, OHNS, and HDS which perform excellent in long runs. However, more durable metal alloys come with more cost. Thus the trick used by majority of mould makers is using harder and durable alloys in most critical areas i.e. the core inserts, cavity inserts, cavity plate and stripper plate.

V. Runner design
An effective runner design leads to effective parts, less consumption of time and less wastage of plastic in runners. There are several runner system designs the mould makers can choose as per the requirement. A two cavity mould will require a simple Y-Balance system assuming each part has one gate. The care should be taken that when more than one part is to be moulded, the effective length of runner from sprue should be same for all parts. This will ensure that same time is required for all parts to be filled with same pressure and velocity of melted plastic.

VI. Mould temperature
Mould temperature plays a vital role in solidification stage in moulding. The temperature control of mould is hence necessary and controlled with care. More than required temperature will result in poor solidification with deformed part at ejection. Too low temperature will solidify the plastic in the runner system much prior it reaches the cavity. This will not only be good for part, but will also be bad for the moulding machine as it will create excess pressure in the barrel and on the screw plunger.
Mould temperature control depends on type of moulding as well. Injection moulding requires mould cooling whereas compression moulding requires mould heating. For cooling, air cooled moulds are most conventional and cheap method but is in affective for large moulds. Water cooled system are used in large moulds with much improved performance than that of air cooled but requires complex channels carved inside the moulds which is time consuming and costly.

VII. Air vent location and size
Air vent is a necessary element in the mould. The air trapped in cavity needs to be rejected as the cavity gets filled with molten plastic. It is seen in practice that air gets trapped in an area far away from gate location. This is due to the air compressibility. Molten plastic pushes the air towards the far end and cannot

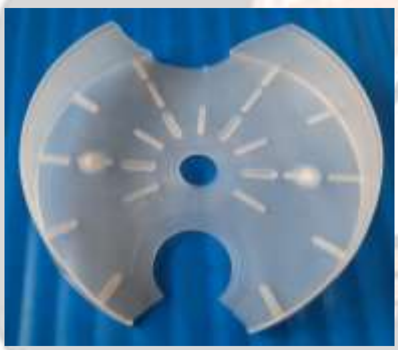
compress it further causing bubble. Thus air vents are located in such areas if cavity is large. In small moldings, the "clearance fit" provided on ejection rods is sufficient to let the air escape. The gap should be in a range of 0.02 to 0.03 mm.

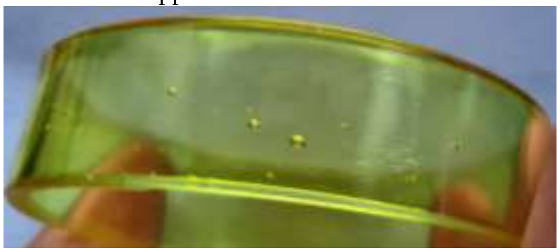
VIII. Mould opening/closing speed
This is the speed of mould operation. If the mould closing speed is too high, the collision of plates will affect the part quality and will also lead to wear and tear of mould itself. If the opening speed is too high, there will be fluctuations in quality and also a high probability of surface scratches on part. Modern moulding machines are programmed considering this and have the speed variation in three stages. In first stage the speed is slow, followed by a fast second stage and again slow stage at end.

5. DEFECTS FOUND IN INJECTION MOULDING PROCESS AND THEIR REMEDIES

There are many defects that can be encountered while moulding. This is because the process has many variables that have implications on the final part quality. It's hence important to know the potential defects and the way to eliminate or at least reduce them. Following some defects that can be usually seen and can be avoided by some easy tricks and techniques are discussed.

Table -1 Defect and their remedies in moulding.

Sr. no	Defect	Remedy
1	Short shot 	Increase injection pressure or injection time. Increase shot size by proper calculation
2	Flow lines 	Increase injection temperature or decrease injection pressure. Change gate location if jet is abruptly gets stopped in cavity
3	Flash 	Decrease injection pressure or change mould design to have less gaps in cavity

4	Blow holes/ air bubble trapped 	Increase injection pressure or add air vents in mould
5	Warpage 	Induce slow cooling of part after solidification or place weight on parts to reduce the effect.
6	Surface scratches 	Apply lubricant spray on cavity surface to reduce the effect.
7	Distortion 	Increase solidification time or decrease mould temperature. Reduce ejection speed and ejection pressure
8	Colour inclusion 	Proper purging of plastic after production

9	Thermal cracking 	Usually occurs in initial cycles, increase mould temperature
10	Blow holes 	Proper air vents required. Slow down screw rotation speed to reduce air inclusion

6. LATEST TRENDS IN INJECTION MOULDING

I. IML (In Mould Labeling)

In mould labeling is a part of In-Mould Decoration process which eliminates labeling time and the process after the moulding is done. The process is carried out by placing the labels to be put on part placed inside the cavity even before plastic is injected. The process is suitable for mass production. The most common example is: - Ice-cream boxes with labels on all four sides.

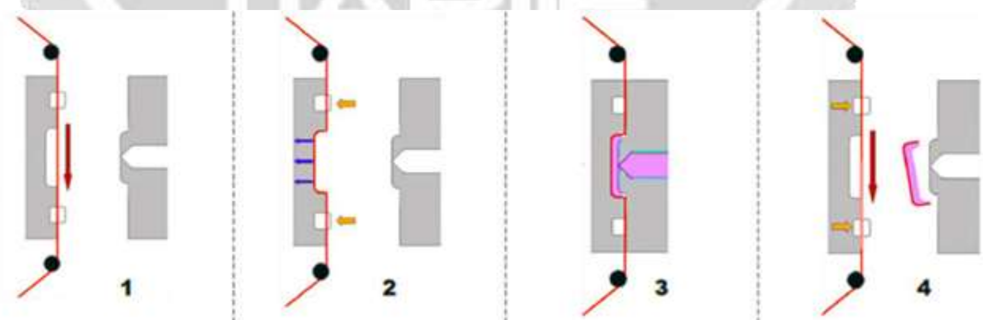


Fig -6 In-Mould Labeling Process

II. Stack Moulds

This is advancement in conventional injection moulds. Conventional mould has only one mould face while stack moulds have number of mould faces. The mould is divided into three parts. One side is the injecting end which has sprue bush in it. The central core has two faces and both faces have cavities. And third side is a supporting part which is used to open and close the mould.

One other type of stack mould has a rotating central core. This core has four moulding faces which rotate around its central axis. Due to this rotation, a new face is exposed to the injecting side after a cycle is completed. This increases the productivity as no time is wasted in removing the part. Then next cycle is underway, previous part is already facing away from injecting end thus part can be removed without stopping the machine for long time.

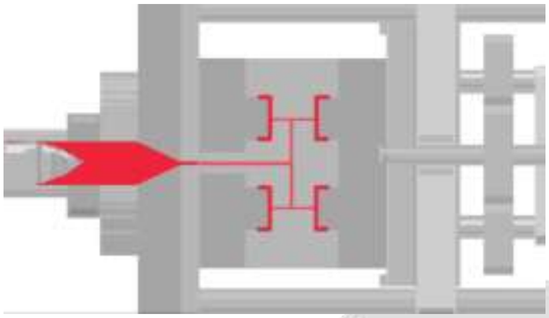


Fig -7 Stack Mould – Mould Closed – Injection Done

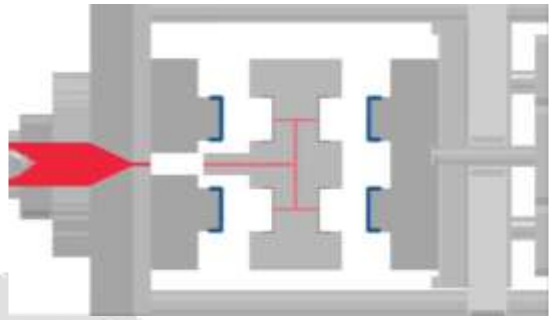


Fig -8 Stack Mould – Mould Open – Part Ejection

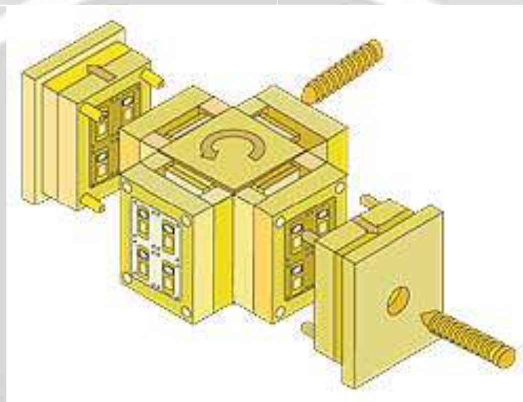


Fig -9 Rotating Cube Stack Mould

III. Micro Moulding

Micro-molding is a highly specialized manufacturing process that produces extremely small, high-precision thermoplastic parts and components with micron tolerances. The process starts in a tooling department where a mold is created that has a cavity in the shape of the part desired. Thermoplastic or resin is rapidly injected into the cavity, creating the component or part at high speed

Part sizes in micron scale that too in high precision and accuracy is possible in Micro Moulding. Part dimensions below 1mm with complex shapes and intricate patterns are maintained by this process. This process can be single shot or two shot. Two shot micro moulding refers to use of two types of plastic injected into cavity at same time.

IV. Bio Degradable Plastics in Injection moulding [2]

Traditional plastics do not decompose easily or rather take years to do so. Biodegradable plastic is a variety that can decompose naturally in the environment. Fermentation of canola oil or sugar can produce a polymer that decomposes under the right temperature and humidity conditions.

This makes biodegradable plastic especially well-suited for items and products which will be discarded after one use.

The most common types of biodegradable plastic resins used in plastic injection molding include:

1. Thermoplastic Starch-based Plastics (TPS)
2. Polyhydroxyalkanoates (PHA)
3. Polylactic Acid (PLA)
4. Polybutylene Succinate (PBS)
5. Polycaprolactone (PCL)

V. Gas Assisted Moulding [3]

The basic concept of the gas-assisted molding process is quite similar to regular injection moulding process. In this gas-assisted molding, the plastic material is melted with the help of extruder barrel and injected into the mold cavities like the regular injection moulding process but only up to 70%~80% of the mold volume. Plastic melts in contact with the mold walls begins to solidify and nitrogen gas is injected into the mold through strategically designed and placed gas channels, providing pressure that pushes the plastic material into the mold extremities and finally the molded part is ejected like the regular injection molding process.

VI. Moulding of Structural Foam [4]

The Structural Foam Process is a low pressure injection molding process where an inert gas is introduced into melted polymer for the purpose of reducing density and hence weight of the finished product. Structural foam molded products have cellular cores surrounded by rigid, integral skins. Foaming agent (NI, CO2 or CBA) is introduced into the polymer melt stream, creating a homogenous mixture of polymer and gas. The mixture is short-shot injected through nozzles into the mold in a volume that is less than the amount required to mold a solid part. Injection pressure and expansion of the polymer/gas mixture fills the mold. A porous skin is formed when the melt contacts the cold surface of the mold. The expanding polymer/gas mixture forms the cellular core.

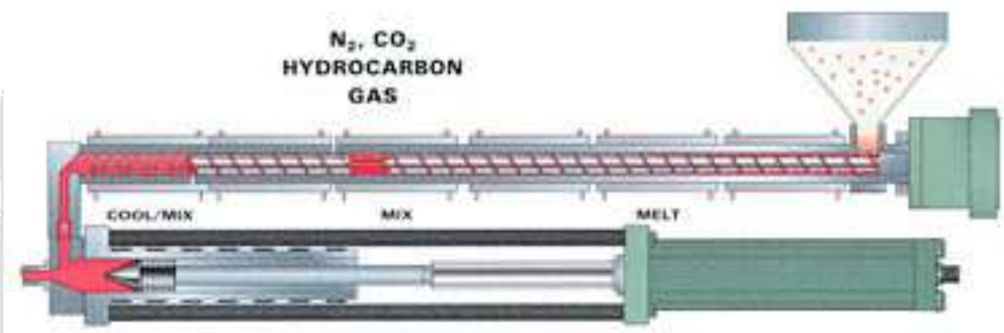


Fig -10 Injection of Structural Foam Along With Plastic

VII. Use of Robotics In Injection Moulding [5]

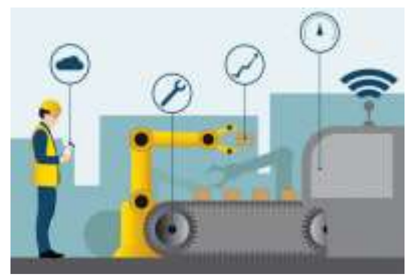
Robots are made to do things which are either too difficult for humans or too labour intensive and time consuming. Robots can handle these chores easily and relieve worker's extra work which in turn improves productivity. Highly repetitive actions are tedious to do and also can cause fatigue to workers. This is why robotics in injection moulding process can be pain reducer. Machine tending – loading and unloading of moulded parts from machine to further processes is the best area for robotic automation. In market there are various types of robots available from which a few are discussed below.

1. **3-Axis Robots | Top Entry Robots**
   Part picking, In-Mold Decorating (IMD), In-Mold Labeling (IML), Insert Loading, Inspection, Stacking and Palletizing are some of the processes that can be done with this type of robot. These are used on Horizontal Plastic Injection Molding Machines (top entry automation applications) and applications requiring higher speed.
2. **6-Axis Robots | Articulated Robots**
   Flexible automation on both Horizontal and Vertical Plastic Injection Molding Machines is done with these robots. Larger working envelope and flexibility in upstream and downstream processes are the advantages of Articulated Robots.
3. **Collaborative Robots (Cobots)**
   The name itself suggests the function of the robot. Collaborative work of robot with humans is the primary advantage of this type. Workers can work safely alongside these robots. Also programming of these robots is way simpler than traditional robots. While traditional robots require an engineer to write the codes for a certain movement to be done, cobots can be easily programmed just by moving the arm in desired location and it will remember the movement. This not only makes working alongside easier but also safer.

4. **4-Axis SCARA Robots**
   Selective Compliant Assembly Robot Arm - 4-axis robots typically work in Plastic Injection Molding automation applications for loading and unloading Vertical Injection Molding machines. They have a circular work envelope with flexibility. These can work with other robots such as 3 axis or 6 axis robots for work sharing and high precision automation.
5. **Side Entry Robots**
   Side entry robots are single axis robots that perform a specific application. They are capable of extreme high speed applications such as placing decorations / labels for In-Mold Decorating (IMD) / In-Mold Labeling (IML) and part removal.
6. **Servo Sprue Pickers**
   Servo Sprue Pickers are simple pick-and-place robotic devices used for sprue removal or part and runner separation in plastic injection molding automation. All three axes i.e. X, Y, and Z axis are servo driven for maximum flexibility and programmability. A simple hand held pendant is enough to move the robot in desired location with great flexibility and simplicity. Servo sprue pickers are also capable for pre-staging above the mold. This minimizes cycle time. They are flexible allowing for deposit on both sides of the injection molding machine - runners out on one side and parts out on the other.

1). 3-Axis Robots | Top Entry Robots

2). 6-Axis Robots | Articulated Robots



3). Collaborative Robots (Cobots)

5). Side Entry Robots

6). Servo Sprue Pickers

4). 4-Axis SCARA Robots

VIII. Hot Runner Injection System

Hot Runner System is a subtype of Runner-less Moulds. This system is effective in eliminating the use of cold runners which are then required to trim after moulding. This not only saves the material from wasting but also saves extra effort and time required after moulding is done. Most modern mass production houses use Hot Runner system as it reduces effort and time even though it requires more initial cost. Advantages of Hot Runner system are – shorter cycle time, easier to use, lesser sing marks, flexible design options and balanced melt flow. Automation in manufacturing is possible as no after trimming is required.

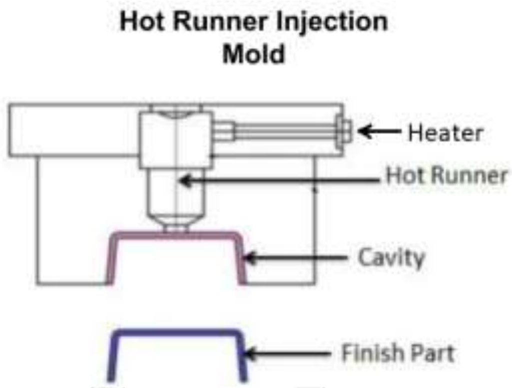


Fig -11 Hot Runner System Schematic

IX. Injection – Compression Moulding (ICM)

As the name suggests, this process is conjunction of two moulding processes namely injection and compression moulding. This is a special application moulding process. Usually insert moulding (Moulding of a part with a preformed component – often metal component – loaded into cavity where it is then overmoulded with plastic) is carried out. These inserts are to be placed such a way that they do not fall before moulding thus this process is preferred. This process requires special machines and mould design. Also there is no runner system expected hence usually has Hot Runner System employed.

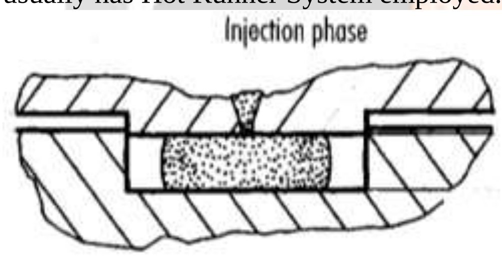


Fig -12 Injection of Plastic Material in Cavity

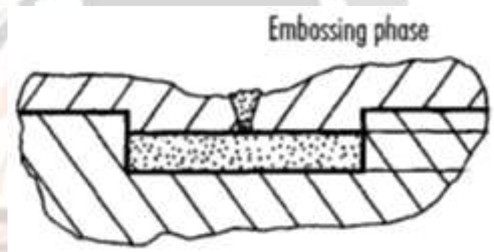


Fig -13 Compression of Plastic in Cavity

X. Collapsible Core Moulding

For Internal threads or to induce an undercut in a part, without using split moulds, collapsible cores are used. These cores operate such that internal threads or any other features that are to be moulded on inner surface of part has becomes much easier. The faces expand when an actuator rod is inserted and collapse when it is removed.

Fig -14 Collapsible Cores

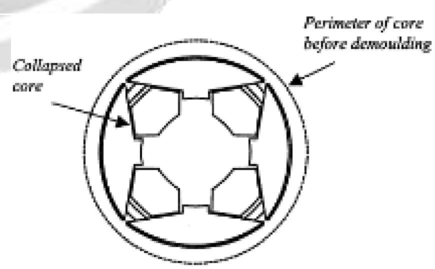


Fig -15 Schematic of Collapsible Core

7. CONCLUSIONS

To produce any plastic component in less time while not compromising with quality, Injection Moulding Process should be implemented. Numerous combinations of surface finish, material, shape, size, and colour can be made possible by this process.

Many organizations have started using non conventional mould designs such as stack moulds and automation techniques such as robots to increase production. Modern analysis tools are also used before the actual process is underway. This benefit in accurate representations of processes and corrective operations needed. This also helps in saving large finances and time which were consumed during conventional trial and error method.
By controlling the process parameters, mould design parameters and using latest trending methods mentioned above, the process can be used at its best.

7. ACKNOWLEDGEMENT

I acknowledge all the staff, employees and management of Amol Enterprises, MIDC Satpur, Nashik for their kind support and guiding me to be capable enough to complete this paper.

8. REFERENCES

[1] https://en.wikipedia.org/wiki/RAL_colour_standard
[2] https://www.stelray.com/biodegradable-plastics-for-injection-plastic-molding/
[3] https://dienamics.com.au/blog/what-is-gas-assisted-injection-moulding/#:~:text=The%20basic%20process%20of%20Gas,mould%20is%20about%2070%25%20full.
[4] https://www.wilmingtonmachinery.com/structural-foam-injection-molding-process/
[5] https://www.rnaautomation.com/blog/robotics-in-injection-moulding/

Books Referred:-

[1] Dominick V. Rosato, Donald V. Rosato, “Injection Molding Handbook”, CBS Publishers & Distributors, ISBN 8123913648, 9788123913643, year – 200
[2] Georg Menges, Walter Michaeli, Paul Mohren, “How to Make Injection Molds”, Hanser Publishers, Munich, eISBN: 978-3-446-40180-8 Print ISBN: 978-3-446-21256-5, year - 2001

Image References:-

[1] Fig -1 https://is2.ecplaza.com/ecplaza2/products/b/bb/bb8/1092135576/supla-synthetic-raw.jpg
[2] Fig -3 http://haitianpm.com/en/products/haitian-mars-series-ma/#ma5300iii-front
[3] Fig -4 https://image.made-in-china.com/2f0j00TnUEDiWFverQ/Plastic-Pellets-Hopper-Dryer-for-Injection-Molding-Machine.jpg
[4] Fig -5 https://www.plasticmoulds.net/ejection-system-plastic-mold.html
[5] Fig -6 https://www.arion.asia/wp-content/uploads/2020/10/IMD-Process.jpg
[6] Fig -7, 8 https://www.evcoplastics.com/processes/stack-molds
[7] Fig -9 https://www.plasticstoday.com/new-turns-multicomponent-mold-revolution
[8] Fig -10 https://www.wilmingtonmachinery.com/structural-foam-injection-molding-process/
[9] Fig -11 https://www.smlease.com/wp-content/uploads/2017/11/Hot-Runner-Vs-Cold-Runner-System.png
[10] Fig -12, 13 Georg Menges, Walter Michaeli, Paul Mohren, “How to Make Injection Molds”, Hanser Publishers, Munich, eISBN: 978-3-446-40180-8 Print ISBN: 978-3-446-21256-5, year – 2001, page no:- 594
[11] Fig -14 https://cdn.genalpha.com/2/images/rt-cores-cat.jpg
[12] Fig -15 https://ars.els-cdn.com/content/image/1-s2.0-S0890695599000966-gr4.gif

BIOGRAPHY

Mayuresh Nitin Madiwale

Graduate Mechanical Engineer,
Intern at Amol Enterprises, MIDC Satpur Nashik