CHAPTER 3

CONCEPTS OF INJECTION MOULDING

Injection moulding is a special manufacturing technique used to make parts from plastic materials. In order to accomplish this, the molten plastic is injected into a mould at a high pressure. The mould it is injected into is the inverse design of the desired shape in order to produce the shape in the way it needs to be designed. The mould used in the injection moulding process is made from a mould maker, or toolmaker. The mould itself is made of metal, which is usually either aluminum or steel. It is then precision-machined in order to create all of the features needed to form the part in the way it is needed. This process is used to create very tiny components for items such as cell phones to large items, such as the entire body panel used for a car. Other common items made with injection moulding include outdoor furniture and bottle caps.

3.1 INTRODUCTION

A great deal of time and thought must be put into the creation of a mould for injection moulding. Besides achieving the desired shape, the mould needs to be made in a manner that will prevent the product being created from being trapped in the mould. In addition, care must be made to ensure the mould will be able to be completely filled with the molten resin before it solidifies. Care must also be made to compensate for material shrinkage and to reduce the chances of imperfections being created in the final product. The raw material used in injection moulding is called resin. Usually, it is in pellet form first and is melted by force and heated before it is injected into the
mould. The plastic then flows toward the chamber and solidifies. This forms is known as the attached frame. The frame is made of the sprue and runners. The sprue is the main channel the plastic flowed through from its molten resin reservoir. The runners are used to transport the molten resin to the gates, which are the points of injection. The sprue and runner is cut off and thrown away after the product is produced. Some moulds are even designed to automatically remove this excess plastic. In order to extract the product from the mould, the mould needs to be able to be separated into at least two halves. These halves are called the cavity and the core.

In this way, the product can be more easily extracted. Since the mould needs to be pulled apart in order to remove the product, care must be taken in the creation of the mould to ensure it will not be "locked in." In other words, the sides of the objects usually run parallel to the area where the core and cavity become separated. The ultimate quality of the plastic product created from injection moulding depends on the quality of the mould used to create it. In addition, the care taken to complete the injection moulding process accurately and thoroughly also has an impact on the quality of the product. It is highly important for the molten resin to be at the right pressure and temperature during the injection moulding process in order to get the desired results because this helps the resin flow better to every part of the mould. The various parts of the mould also must come together precisely. If they do not, small leakages will occur. When this occurs, it is referred to as flash. To prevent flash from occurring, which can cause damage to the mould, technicians slowly increase the pressure used in new moulds until they determine the proper amount of pressure to get the desired results. Other factors, such as temperature, venting, and resin moisture, can also cause flash to occur. Making moulds is generally an expensive undertaking. Therefore, it is rarely used when thousands of products need to be produced. Steel moulds
are usually more expensive to create than aluminum moulds, but they last much longer and are worth the investment in the long run because more products can be made with one mould.

Injection moulding is the most commonly used manufacturing process for the fabrication of plastic parts. A wide variety of products are manufactured using injection moulding, which vary greatly in their size, complexity, and application. The injection moulding process requires the use of an injection moulding machine, raw plastic material, and a mould. The plastic is melted in the injection moulding machine and then injected into the mould, where it cools and solidifies into the final part. Factors that affect the quality of moulded parts can be grouped into: part design, mould design, machine performance and processing conditions. The steps in this process are described in greater detail in the next section.

Injection moulding is a manufacturing process which is 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 mould 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.

In recent years, the communication products like cell phone are widely applied throughout the world. The designs of cell phone have a tendency to be thin, light and small and more convenient style. Therefore, the shapes of smart cell phone are changing, and more features have to be tightly packed into smaller volumes within the housing. In order to procure more
space for the tightly packed components inside, the wall thickness of the housing parts must originally be reduced to less than 3 mm in thickness. Plastic injection moulding is one of the most important procedures applied for forming an injection-moulded thermoplastic part with a thin-shell feature. Machining parameters, moulding material, product and mould designs are major factors affecting the quality of thermoplastic parts produced by injection moulding.

3.2 WORKING PRINCIPLE

In this process, the plastic materials usually in the form of powder or pellets are fed from hopper into the injection chamber. The “piston and cylinder” arrangement is used to forward the material inserted from the hopper in to the injection chamber. The plastic material is heated in the injection chamber with the application of heating elements.

![Injection moulding setup](image)

**Figure 3.1 Injection moulding setup**

The cooling system is also used to maintain the temperature of the injection chamber. The molten plastic material is then injected into the mould cavity through a nozzle. The moulded part is cooled quickly in the mould. Thereafter, the final plastic part is removed from the mould cavity. The
process cycle for injection moulding is very short, typically between 2 to 60 seconds. Figure 3.1 shows the injection moulding setup. The complete injection moulding process is divided into four stages: clamping, injection, cooling and ejection.

3.3 CONSTRUCTION

3.3.1 Clamping

The two halves of the mould must be tightly closed, before the molten plastic material is injected into the mould. One half of the mould is attached to the injection unit (nozzle) and other half is allowed to slide on the guide ways. The clamping of mould is operated hydraulically which it pushes the moving half part of the mould towards the fixed part to make an air tight chamber. The force and the time required closing and open the mould depends upon the machine capability.

3.3.2 Injection

During this process, the plastic material is melted by the application of heat and forwarded through the piston towards the nozzle and finally into the mould. The function of Torpedo in the heating zone is to spread the molten plastic into the thin film. The molten plastic is then injected into the mould cavity quickly. The amount of material that is injected into the mould is referred to as the shot volume. The injection time can be estimated by the shot volume, injection power and pressure. The schematic of injection moulding process is shown in Figure 3.2.
Injection moulding machines can fasten the moulds in either a horizontal or vertical position. The majority of machines are horizontally oriented, but vertical machines are used in some niche applications such as insert moulding, allowing the machine to take advantage of gravity. Some vertical machines also don't require the mould to be fastened. There are many ways to fasten the tools to the platens, the most common being manual clamps (both halves are bolted to the platens); however hydraulic clamps (chocks are used to hold the tool in place) and magnetic clamps are also used. The magnetic and hydraulic clamps are used where fast tool changes are required.

The plastic injection moulding process produces large numbers of parts of high quality with great accuracy, very quickly. Figure 3.3 shows the working principle of the injection moulding machine. Plastic material in the form of granules is melted until soft enough to be injected under pressure to fill a mould. The person designing the mould chooses whether the mould uses a cold runner system or a hot runner system to carry the plastic from the injection unit to the cavities.
A cold runner is a simple channel carved into the mould. The plastic that fills the cold runner cools as the part cools and is then ejected with the part as a sprue. A hot runner system is more complicated, often using cartridge heaters to keep the plastic in the runners hot as the part cools. After the part is ejected, the plastic remaining in a hot runner is injected into the next part.

### 3.4 PROCESS OF INJECTION MOULDING

Just as in die casting, the mould is specially made for each part, and the basic elements of each mould are the same, including sprue gates runners and vents. In addition, the location of ejection pins is usually specified in the mould design, since this points have visible marks (therefore ejection is usually done from the core side, and is usually mounted into the mould half mounted on the moving platen). The cavity is divided between the two mould halves in such a way that the natural shrinkage of the moulding causes the part to stick to the moving half. When the mould opens, the ejector...
pins push the part out of the mould cavity. The different types of mould is
given below.

3.4.1 Two-Plate Mould

This consists of two halves fastened to the two platens of the
moulding machine's clamping unit. When the clamping unit is opened, the
mould halves separate. Moulds can contain one multiple cavities to produce
one or multiple parts in a single shot (Figure 3.4). The parting surface is the
surface shared by the two mould halves. The cooling system is made up of
passages in the mould that are connected to an external pump. Water is
circulated through them to remove heat from the hot plastic. The air trapped
in the cavity passes through the small ejector pin clearances in the mould and
through narrow vents that are machined into the parting surface (typically
about 0.03 mm deep and 12 to 25 mm wide).

Figure 3.4. (a),(b) Two-plate mould
3.4.2 Three-Plate Mould

The molten plastic flows through a gate located at the base of the cup-shaped part, rather than at the side. This allows more even distribution of melt into the sides of the cup. In the side gate design in the two-plate the plastic must flow around the core and join on the opposite side, possibly creating a weakness at the weld line. Secondly, the three-plate mould allows more automatic operation of the moulding machine. As the mould opens, the three plates separate, this forces the runner to break from the parts, which drop by gravity or using air-blower into collecting containers put under the mould.

![Figure 3.5 Three-plate mould](image)

3.5 MOULDS WITH CORES/CAMS

Many injection moulded parts have some part of the geometry that is inaccessible to either of the mould halves. Such regions must be created by means of extra moving parts in the mould. Figure 3.6 shows a cup-shaped part with a through-hole. One method to mould this part is by the use of a core.
Figure 3.6 Thermo Plastic Injection Moulding Schematic

The Figure 3.7 shows the steps of the mould opening. As the mould opens, the green part is forced to slide to the right. The mould-piece that creates the insert geometry is attached to the green piece by the blue bolt.

Figure 3.7 Injection moulding a part with side action using core
3.5.1 Cooling

The injected molten plastic begins to cool as soon as it comes in contact with the mould surfaces. As the moulded part cools, it will solidify into the desired shape of the product.

3.5.2 Ejection

The moulded part, which is attached to the rear half of the mould has to be ejected from the mould. When the mould is opened, an ejector mechanism is used to push the part out of the mould. Force must be applied to eject the plastic part because during cooling the moulded part shrinks and adheres to the mould surface. A mould release agent should be sprayed onto the mould surfaces prior to injection of the material. Once the moulded part is ejected, the mould will be closed for the next shot to be injected. The important process parameters that have to be considered during the injection moulding process are injection temperature and pressure, shot volume, mould temperature, cooling time, ejection temperature, and cycle time. Some of the common injection moulding defects are flash, blister, warping, bubbles, unfilled sections, jetting, sink and ejector marks. These defects can be eliminated by optimal selection of the process parameters.

3.6 MATERIALS USED

The injection moulding process can be used to process materials such as

- Acetal, Acrylic
- Acrylonitrile Butadiene Styrene (ABS)
- Cellulose Acetate, Polyamide (Nylon)
• Polycarbonate, Polyester
• Polyether Sulphone (PS)
• Polyetheretherketone (PEEK)
• Polyetherimide, Polyethylene
• Polyphenylene Oxide
• Polyphenylene Sulphide (PPS)
• Polypropylene (PP)
• Polyvinyl Chloride (PVC)
• Elastomers

3.7 TYPES OF INJECTION MOULDING MACHINES

There are several types in the injection machine, and the difference is made by how these two devices are arranged.

a. Horizontal injection machine : both mould clamping device and injection device compounded horizontally

b. Vertical injection machine : both mould clamping device and injection device compounded vertically

c. Two-colour injection machine

d. Rotary injection machine

e. Low foam injection machine
f. Multi material injection machine

g. Sandwich injection machine

3.8 SELECTION OF INJECTION MOULDING MACHINE

The selection of injection moulding machine is carried in the following aspects

- Select by injection volume
- Select by mould clamping pressure
- Nozzle structure
- Injection mechanism
- Backflow prevention ring
- Drying machine

3.9 INJECTION MOULDING CONSIDERATIONS

3.9.1 Steps in Designing an Injection Mould

The design of injection mould tooling requires several steps. The moulding directions will determine the number of inserts/cams required, which severely affects the cost of the tooling. After finding the suitable moulding direction, the parting lines are determined. The parting planes form the surface of the mould halves – usually, the parting planes are formed by extending the parting line outwards, perpendicular to the moulding direction. The gating design determines where to locate the gate(s). If a multiple cavity mould is made, the relative positions of the multiple parts are determined.
The runners are designed, and spruce is located. Then the functional parts of the mould are created next – this includes the part ejection system, systems to eject the solidified runners etc. Finally, the alignment rods that will keep all mould components aligned during operation are designed and Figure 3.8 shows a cup-shaped part which has parting line is present inside and parting plane is present outside.

![Figure 3.8 sample part](image)

**Figure 3.8 sample part**

![Figure 3.9 Stages of development of the injection mould](image)

**Figure 3.9 Stages of development of the injection mould**

The ideal parting surface for this line is a plane. Figure 3.9 and 3.10 shows the stages of development of the mould. Note that here a plate is used to eject the cup, rather than ejection pins.
3.9.2 Considerations in Design of Injection Moulded Parts

There are several geometric and design considerations for parts manufactured using injection moulding. Many of these considerations have resulted in a large set of guidelines for geometric features in the design. The two biggest geometric concerns are (i) proper flow of the plastic to all parts of the mould cavity before it solidifies, and (ii) shrinking of the plastic resulting in sink holes. Some examples of the first concern include: if the part thickness is too small, plastic flow is restricted due to high friction; if the gate is too far away from some small features of the geometry, or is there is a constriction in the path along which the plastic will flow; another guideline is that the cross section of the part should not change abruptly, since this leads to poor flow. Regarding the second concern, guidelines include (a) maintaining uniform cross-section thickness throughout the part; use of ribs and gussets to provide mechanical strength instead of using thicker sections etc. Figure 3.11 shows a comparison of strength of parts of the same length but different cross-sections – the examples demonstrate that the use of ribs will result in lighter parts for a
given stiffness. Figures below demonstrate some of the guidelines for common moulded geometry.

<table>
<thead>
<tr>
<th>Geometry</th>
<th>Cross-Section Area (square inches)</th>
<th>Max. Stress (psi)</th>
<th>Max. Deflection (inches)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Original Section</td>
<td>0.0600</td>
<td>6250</td>
<td>0.694</td>
</tr>
<tr>
<td>Original Section with Rib</td>
<td>0.0746</td>
<td>2273</td>
<td>0.026</td>
</tr>
<tr>
<td>Thick Section</td>
<td>0.1793</td>
<td>699</td>
<td>0.026</td>
</tr>
</tbody>
</table>

**Figure 3.11** Comparison of breaking stress under bending moments on bars of different cross-section geometry

**Figure 3.12** Example of improving design for injection moulded parts
The main ejection-related guideline is the same as for all casting/moulding processes: providing for tapered shapes by applying a draft angle to all part surfaces with normal vector perpendicular to the moulding direction. Another consideration in injection moulding is regarding weld lines. As the plastic flows to fill the mould, there are some regions where the advancing front of the plastic flowing from different directions meets. At this point the two flows mix together – however, the solidified plastic has slightly less strength along the line where this occurs. Lines along which this happens are called weld lines. An example is shown in the moulding of a phone cover plate.

3.9.3 Optimization

Optimization is the act of obtaining the best result under given circumstances. In design, construction and maintenance of any engineering system, engineers have to take many technological and managerial decisions at several stages. The ultimate goal of all such decisions is either to minimize the effort required or to maximize the desired benefit. Since the effort required or the benefit desired in any practical situation can be expressed as a function of certain decision variables. Optimization can be defined as the process of finding the conditions that give the maximum or minimum value of function. It can be seen from Figure 3.13 for sample factor of production rate.
Evolution is an astonishing problem solving machine. The force working for evolution is an algorithm, a set of instructions that is repeated to solve a problem. The algorithm behind evolutions solves the problem of producing species able to thrive in a particular environment. Placement and routing are two search intensive tasks. Even though agent objects use knowledge to reduce search time, a great deal of searching is still necessary. A good proportion of this search time will be spent optimizing the components’ placement in the layout. In searching for optimum solutions, optimization techniques are used and can be divided into three broad classes, as shown in Figure 3.14. Numerical techniques use a set of necessary and sufficient conditions to be satisfied by the solutions for optimization problem. They sub divide in to direct and indirect methods. Indirect methods search for local extremes by solving the usually non-linear set of equations resulting from setting the gradient of the objective function to zero.

Figure 3.13 Minimum and maximum production rate
3.10 SUMMARY

Injection moulding represents one of the most important processes in the mass production of manufactured plastic parts with complex geometries. The quality of the injection mouldings depends on the material characteristics, the mould design and the process conditions. Defects in the
dimensional stability of the parts result in shrinkage. Severe shrinkage leads to deflection of war page in moulded parts as well as negatively influences the dimensional stability and accuracy of the parts. Many factors including materials selection, part and mould design, as well as injection moulding process parameters can affect shrinkage behaviour in an injection moulded part.

Common quality problems or defects that come from an injection moulding process include voids, surface blemish, short-shot, flash, jetting, flow marks, weld lines, burns, and war page. The defects of injection moulding process usually arise from several sources, which include the pre-processing treatment of the plastic resin before the injection moulding process, the selection of the injection-moulding machine, and the setting of the injection moulding process parameters.

This work attempts to describe the optimization of the injection moulding process parameters for optimum shrinkage performance of a plastic head light of Tata Magic which is made from Polyypropylene polymer. In this the process parameter such as Injection temperature, Injection pressure, Packing pressure and Packing time has been taken to get best combination to optimize the process. The trial-and-error process is costly and time consuming thus not suitable for complex manufacturing processes. In order to minimize such defects in plastic injection moulding, design of experiment, the Taguchi method is applied. Signal-to-Noise ratio was used to obtain the optimal set of process parameters.