CHAPTER 2

INJECTION MOULDING PROCESS

Injection moulding is the most widely used polymeric fabrication process. It evolved from metal die casting, however, unlike molten metals, polymer melts have a high viscosity and cannot simply be poured into a mould. Instead a large force must be used to inject the polymer into the hollow mould cavity. More melt must also be packed into the mould during solidification to avoid shrinkage in the mould. The injection moulding process is primarily a sequential operation that results in the transformation of plastic pellets into a moulded part. Identical parts are produced through a cyclic process involving the melting of a pellet or powder resin followed by the injection of the polymer melt into the hollow mould cavity under high pressure.

2.1 INJECTION MOULDING MACHINE

An injection moulding machine produces components by injection moulding process. Most commonly used machines are hydraulically powered in-line screw machines, although electric machines are appearing and will be more dominant in the market in near future.

The main units of a typical injection moulding machine are the clamping unit, the plasticizing unit, and the drive unit; they are shown in Fig. 2.1. The clamping unit holds the mould. It is capable of closing, clamping, and opening the mould. Its main components are the fixed and moving plates, the tie bars and the mechanism for opening, closing and clamping.

The injection unit or plasticizing unit melts the plastic and injects it into the mould. The drive unit provides power to the plasticizing unit and clamping unit.
Injection moulding machines are often classified by the maximum clamp force that the machine can generate. This is the force that pushes the two mold halves together to avoid opening of the mould due to internal pressure of the plastic melt in the mould. The clamping force of typical injection moulding machines range from 200 to 100,000 kN.

![Injection Moulding process](source: Jonathon website (81))
2.2 THE INJECTION MOULDING CYCLE

There are three main stages in the injection moulding cycle; stage 1, injection, followed by stage 2, holding pressure and plasticating, and finally, stage 3, ejection of the moulded part. When stage 3 is completed, the mould closes again and the cycle starts over again.

2.2.1 Stage 1

INJECTION OF THE PLASTIC MELTS INTO THE MOULD: In stage 1, the mould is closed and the nozzle of the extruder is pushed against the sprue bushing of the mould. The screw, not rotating at this point, is pushed forward so that the plastic melt in front of the screw is forced into the mould.

2.2.2 Stage 2

HOLDING PRESSURE AND PLASTICATING: When the mould is completely filled, the screw remains stationary for some time to keep the plastic in the
mould under pressure, this is called the “hold” time. During the hold time additional melt is injected into the mould to compensate for contraction due to cooling. Later, the gate, which is the narrow entrance into the mould, freezes. At this point the mould is isolated from the injection unit. However, the melt within the mould is still at high pressure. As the melt cools and solidifies, the pressure should be high enough to avoid sink-marks, but low enough to allow easy removal of the parts.

During the plastication stage, the material is pushed forward from the feed hopper through the barrel and towards the nozzle by a rotating screw. When the gate freezes, the screw rotation is started. The period of screw rotation is called screw “recovery”. The rotation of the screw causes the plastic to be conveyed forward. As the plastic moves forward, heat from the electric heater bands along the barrel and shear starts to melt the plastic. At the discharge end of the screw, the plastic will be completely melted. The melt that accumulates at the end of the screw pushes the screw backward. Thus the screw rotates and moves backward at the same time. The rate at which plastic melt accumulates in front of the screw can be controlled by the screw backpressure, that is, the hydraulic pressure exerted on the screw. This also controls the melt pressure in front of the screw.

When sufficient melt gets accumulated in front of the screw, the rotation of the screw stops. During screw recovery the plastic in the mould is cooled, but typically the cooling is not finished by the end of screw recovery. As a result, the screw will remain stationary for some period until cooling is completed. This period is often referred to as “soak” time. During this time additional plastic will melt in the extruder from conductive heating. Also, the melted material will
reach more thermal uniformity, although the soak time is usually too short to improve thermal homogeneity significantly.

2.2.3 Stage 3

**EJECTION:** When the material in the mould has cooled sufficiently to retain its shape, the mould opens and the parts are ejected from the mould as shown in Fig. 2.2. When the moulded part has been ejected, the mould closes and the cycle starts over again.

The different stages can be graphically illustrated as shown in Fig. 2.3. The top bar shows the movement of the extruder screw, the second bar shows the action going on inside the mould and the third bar indicates at what times the mould is open and closed. As can be seen in Fig. 2.3 the major part of the injection moulding cycle is the cooling time required for the plastic in the mould to reduce to a temperature where the part can be removed without significant distortion. The main variable that determines the cooling time is the thickness of the molded part.

![Fig.2.3 Injection Moulding cycle time](image)

[Source: Moulding monitoring website (82)]
2.3 THE MOULD

Each mould, sometimes referred to as a tool, is built to exact specifications of the part or parts required by the customer. The mould typically consists of two mould halves. Usually one mould half contains the cavity and forms the outer shape of the part. This part of the mould is called the cavity side. The other mould half contains a protruding shape and forms the inner shape of the part, this mould part is called the core. When the core is clamped against the cavity, the hollow space that is formed defines the shape of the part to be moulded. The plastic is usually injected into the mould from the cavity side.
The mould cavities are cut to dimensions larger than the desired part dimensions to compensate for the plastic shrinkage which occurs during cooling. The cavity dimensions are equal to the part dimensions plus some shrink factor supplied by the material manufacturer. There are usually two shrink factors given, one for dimensions in the direction of the flow and the other for dimensions perpendicular to the direction of the flow. Estimating shrinkage, however, is not straightforward. It is often difficult to predict the melt flow path in parts with complex geometries and therefore, it is not clear which shrink factor to apply.

Part shrinkage is also influenced by the process conditions.

2.3.1 THE RUNNER SYSTEM

A mould basically consists of properly designed sprue, runner, gate, and cavity. The sprue is the channel, cut in the stationary platen that transports the melt from the plasticator nozzle to the runner. Once the plastic melt enters the mould, it flows through a distribution system, called the runner system, and then through the gates into the mould cavities. In a so-called cold runner system, a new runner is moulded in each moulding cycle and the runner is ejected together with the moulded parts. The plastic of the runner can often be reprocessed and moulded again. In the design of the runner system the objective is to have the plastic reach all gates at the same time. This is an important issue in multi-cavity moulds. In a rectangular runner system, the number of cavities is a multiple of two. In a circular runner any number of cavities can be used.

The gate connects the runner to the actual part. The cross section of the gate is usually small so that the runner can be easily removed from the part and does not leave a large gate mark on the part.

2.3.2 MOULD COOLING
During the moulding cycle, heat is first required to be put into the material and then that heat must be removed as quickly, and as consistently as possible, if the rapid production of consistent products is to be obtained. As most modern injection moulding machines are screw machines, heat input is relatively easy. Heat removal from the plastics material contained in the mould is, however, difficult as plastics material contains a lot of heat and has a low thermal conductivity.

Cooling allows the plastic to solidify and become dimensionally stable before ejection. Heat that has been transferred to the mould by the molten plastic is carried away by a coolant that circulates through cored passages in the mould. Coolant temperature and flow rate determines the efficiency of heat removal. Cooling the moulded components uniformly may mean either, cooling the mould with different flow rates of cooling medium in different areas or, using the same flow rate throughout the mould but with different temperatures of cooling medium. The objective is to cool the components as quickly and as uniformly as possible, while ensuring that defects such as poor surface finish and changes in physical properties are not encountered.

The design of the mould cooling passages also affects the ability to remove heat from the mould. The mould surfaces closest to the cored passages will cool first. Differences in mould temperature distribution will affect reproducibility of moulding parts.

2.3.3 PROCESS VARIABLES

Each process variable can be categorized into one of the five main types such as speed, pressure, time, temperature and stroke. The relationship between the five is of an interactive nature as each variable cannot be readily isolated. This
relationship can be simply demonstrated, for example, upon increasing the hydraulic back pressure, the linear retraction speed of the screw (during recovery) changes causing an increase in the screw recovery time, the melt temperature and/or homogeneity. As a result of the increase in the melt temperature further changes occur to the mould fill time, the injection pressure, the mould temperature, the product ejection temperature and the product dimensions. Hence, by increasing a pressure variable (for example, the hydraulic back pressure) three other main variable types are collectively influenced. More important, the process and subsequently the moulded components are affected.

When changes to a particular process variable or machine setting do occur (which significantly affect the stability of the moulding process so that defective components are produced) it is important that the correct process variable should be controlled so as to rectify the disturbance. For instance, the selection of the wrong hopper throat temperature can cause short mouldings to be produced which then misleads the moulder into altering other variables (for example the holding pressure and/or, the shot volume and/or, the mould filling speed etc.) to overcome the short moulding problem. As the initial selection was incorrect, the process remains unstable but, in changing another variable type, the moulder is led to believe that the problem is resolved. However, in reality defective and/or inconsistent parts will continue to be produced throughout the production run.

The following headings highlight typical process variables which need to be monitored and/or controlled during each cycle. Each of the listed variables will not be discussed in detail to highlight the importance of each variable with respect to the stability of the process. Process variables can be categorized as follows:
2.4 SPEED RELATED PROCESS VARIABLES

Mould opening and closing speeds
Injection speed
Screw rotation speed
Screw recovery speed
Component retraction speed

2.5 PRESSURE RELATED PROCESS VARIABLES

Injection pressure
Holding pressure
Hydraulic back pressure

2.6 TIME RELATED PROCESS VARIABLES

Injection time
Holding pressure time
Pause (dwell) time
Cooling time
Cycle time

2.7 TEMPERATURE RELATED PROCESS VARIABLES

Melt temperature
Mold temperature
Barrel temperature
Cooling water temperature
The extent of the variability encountered is a good indicator of the level of inherent stability present within the moulding process. Hence, the smaller the variation the more consistent and stable is the process.