

INJECTION MOULDING

Introduction

This is a simple process in which the molten polymer is forced into a mould cavity where it solidified. The product is then ejected from the mould cavity.

Advantages

- **The process is accurately repeatable.**
- **Complex designs are possible.**
- **No preheating & preforming of material are required.**
- **No finishing and painting are required.**
- **High product rate is obtainable.**
- **Labour cost is low.**



Disadvantages

Disadvantages

- **Moulds and machines are complex.**
- **Machine and mould costs are high.**
- **The operating method is complicated.**
- **The process deals with high pressure (dangers).**
- **There are more wastes from the process.**
- **Stresses in the finished products are significant.**



Types of injection moulding machines

1. Reciprocating-screw injection moulding systems

2. Plunger injection moulding

<u>Plunger injection moulding</u>	<u>Reciprocating-screw injection moulding</u>
High pressure losses	Low pressure losses
Low shear rates	High shear rates
Poor mixing because of laminar flow	Better mixing because of non-laminar flow



Types of injection moulding machines

<u>Plunger injection moulding</u>	<u>Reciprocating-screw injection moulding</u>
Poor heating (entire process from heater bands)	Better heating (from heater bands and frictional heating)
Less homogeneous melt	More homogeneous melt
Likely to polymer degradation due to longer period of time for heating the polymer	Fast heating time, so less likely to thermal degradation
Less homogeneous melt	More homogeneous melt



Types of injection moulding machines

Plunger injection moulding

Likely to polymer degradation due to longer period of time for heating the polymer

Poor pressure transfer into the cavity due to large variability of the materials being compressed

Reciprocating-screw injection moulding

Fast heating time, so less likely to thermal degradation

Better pressure transfer into the cavity



Types of injection moulding machines

Plunger injection moulding

From above, non-uniform shrinkage and stress in the moulding

Less flexibility and versatility such as plunger design and cleaning

Reciprocating-screw injection moulding

more uniform shrinkage and stress in the moulding

More flexibility and versatility such as various screw design and cleaning



Machine Components

Machine Components

1. Drive unit - Screw drive, injection cylinder.
2. Injection unit - screw, barrel, nozzle, heaters.
3. Mould unit - sprue, runner, gate.

Injection moulding process

1. Melting stage
2. Mould-filling stage
3. Solidifying or cooling stage



Production Cycle in Injection Moulding

— Connection of the injection and mould units

— Screw back I

— Mould closed

— Injection

— Packing

— Screw back II.

— Ejection

— Cycle (3) to (8) is repeated.



Machine design and components

Drive unit

— Turning or rotating the screw

— Opening and closing the mould

— Injecting the melt into the mould

— Pressurising the melt

— Ejecting the moulding and so on.



Screw

Injection unit

- To transport and melt the polymer.
- To carry out any mixing required during the process.
- To generate a stable and homogeneous supply of polymer melt.

A typical single-stage screw consists of three regions:

- The Feed Zone - This section of the screw has the constant deepest channel and its function is to transport the polymer from the hopper to the compression zone.



Injection unit

— Compression Zone or Transition Zone - The depth of the screw channel diminishes, the polymer being compressed. Compression also speeds up the melting process.

— Metering (pumping) Zone: This is the shallowest section of the screw, its functions being to carry out any mixing required.

There are many configurations of screw depending on the type of polymers being processed.



Injection unit

Barrel

- Hard metallic alloy to prevent wear and to withstand relatively high pressures

- Heated by electrical resistance heaters

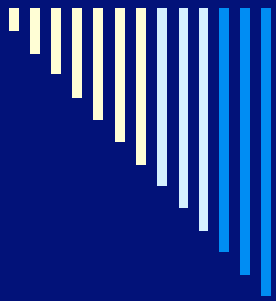
- Expressing by the dimension of the barrel is a ratio of the barrel length to its diameter (L/D).



Temperature control and setting

Typical processing temperatures for commercial thermoplastics include as follows:

LDPE	150-280 °C
HDPE	170-290 °C
uPVC	160-200 °C
PP	200-260 °C
PS	180-300 °C
PMMA	190-240 °C
Nylon 6,6	270-285 °C



Nozzle

Nozzle is a connection between two units, these being injection and moulding units. Two problems usually encountered from the nozzle are drooling and freezing of the nozzle. Two common types of nozzle are:

- **Open nozzle**

- **Shut-off nozzle**

Cushion

- **Cushion is the amount of the molten polymer that is left in the barrel between the nozzle and screw tip at the completion of the filling process**

- **The polymer at the cushion is also for shrinkage.**

- **The typical distance of the cushion is in the range of 1.50 - 3.25 mm.**



Mould unit

Mould unit

The mould cavity receives polymer melt flowing from a nozzle via sprue, runner and gate. The mould is a very expensive part of the injection moulding machine.

Clamping unit

Two methods of opening and closing mould are:

— **Hydraulic cylinder system:** The hydraulic cylinder system needs to be designed for rapid opening and closing.

— **Mechanical toggle system:** This system is actuated by a high pressure and low volume cylinder. Toggle system is normally used on small machine and permits relatively short opening strokes.



Mould unit

The advantages of the toggle system are:

- giving high speed of closing and opening the mould.
- comparatively simple.
- Low cost

The disadvantages are:

- no indication of the clamping force.
- more difficult to control
- Wear on the joint of the toggle system
- Closing/opening forces are small.
- more maintenance is required.



Mould unit

Sprue

The sprue is the first part in the mould unit which receives molten polymer injected from the nozzle.

Runner

It is defined as a flow path between the sprue and the gate. The main function of the runner is to distribute the melt to mould cavities from the sprue region.

Recent development of the runners

Use of heated runners (hot runners)

Sprueless injection moulding



Mould unit

Gate system

Gates should be kept as small as possible in order to:

- obtain rapid cooling and reduce the cycle time.
- allow the moulding to be separated
- increase shear rate as the melt enters the gate, reducing the melt viscosity.

Various types of gate are available.



Mould unit

Parting lines and Venting

Parting line is a point where the mould splits and it leaves an impression.

The venting system is used for removing air from the cavity.

Cooling and heating systems

Moulding ejection

Products are released from the machines by using an ejection system. The simplest way of doing this is to use knockout (ejection) pins which contact the moulded part. The ejection pins are mechanically operated.



Mould unit

Mould temperature

This is important because it can minimise the production cycle. The temperature of the mould used depends on the type of polymer and the geometry, including shape and size, of the moulding.



Mould unit

LDPE	30-50 °C
HDPE	30-50 °C
UPVC	40-50 °C
PP	50-80 °C
PS	30-50 °C
PMMA	40-50 °C
NYLON6,6	90-110 °C



Mould unit

Effects of Processing Variables on Production

1. Melt temperature
2. Mould temperature
3. Mould design
4. Injection speed (injection time)
5. Screw rotation speed
6. Injection pressure
7. Hold-on pressure
8. Back pressure
9. Mould clamping speed
10. Polymer used



Polymer melt flow in a mould

This involves the flow of polymer melts in a mould cavity during a non-steady cooling, the temperature of the melt in the cavity progressively decreasing as the melt flows in (filling process occurring in the mould cavity).

The flow in the mould involves:

1. Fountain flow
2. Multi-layers of the flow
 - Skin layer
 - Shear layer
 - Core layer

Each layer has its own characteristics which control the properties of the moulding.



Parameters affecting the thickness of each layer:

1. Mould temperature
2. Initial melt temperature
3. Flow rate (injection speed)
4. Thermal conductivity of polymers
5. Mould thickness



Polymer melt flow in a mould

This involves the flow of polymer melts in a mould cavity during a non-steady cooling, the temperature of the melt in the cavity progressively decreasing as the melt flows in (filling process occurring in the mould cavity).

The flow in the mould involves:

1. Fountain flow
2. Multi-layers of the flow
 - Skin layer
 - Shear layer
 - Core layer

Each layer has its own characteristics which control the properties of the moulding.



Parameters affecting the thickness of each layer:

1. Mould temperature
2. Initial melt temperature
3. Flow rate (injection speed)
4. Thermal conductivity of polymers
5. Mould thickness



Determination of Freeze Time in Injection

Moulding

- Cooling cycle is the longest period in injection moulding. The aim is to keep the cooling cycle as short as possible.
- Parameters involved in cooling process include:
 - Melt temperature
 - Mould temperature
 - Mould thickness



Determination of Freeze Time in Injection Moulding

- For non-steady heat transfer in one dimension, the determination of freeze time can make use of the Fourier equation as shown below.

$$\frac{\partial^2 T}{\partial x^2} = \frac{1}{\alpha} \frac{\partial T}{\partial t}$$

- T = temperature t = time
- x = distance (related to mould thickness)



Determination of Freeze Time in Injection Moulding

Thermal diffusivity (α) is obtained by:

$$\alpha = \frac{k}{\rho C_p}$$

k = the thermal conductivity.

ρ = the density.

C_p = the specific heat at constant pressure.

Values of thermal diffusivity for polymers are in the range of
 $0.5-1.5 \times 10^{-7} \text{m}^2\text{s}^{-1}$



Determination of Freeze Time in Injection

Mouling

$$T^* = \frac{T_3 - T_2}{T_1 - T_2}$$

$$F_o = \frac{x^2}{t}$$

T1 is the initial melt temperature

T2 is the mould temperature

T3 is the freeze temperature

x is half the specimen thickness

t is the freeze time



Determination of Freeze Time in Injection Mouling

An example

For a low-density polyethylene, the following data are experimentally obtained.

$$T_1 = 188 \text{ }^\circ\text{C}$$

$$T_2 = 28 \text{ }^\circ\text{C}$$

$$T_3 = 110 \text{ }^\circ\text{C}$$

$$\alpha = 1.4 \times 10^{-7} \text{ m}^2\text{s}^{-1}$$

$$x = 2 \times 10^{-3} \text{ m (a specimen thickness is } 4 \times 10^{-3} \text{ m (4mm) being produced)}$$

The above gives

$$T^* = 0.51$$

$$F_0 = 0.38$$

Therefore, the freeze time is 10.8 seconds



Effects of processing conditions on the freeze time, changing one thing at a time:

- Doubling the specimen thickness from 4 mm to 8 mm gives the freeze time of 21.7 seconds (100% increase).
- Doubling the mould temperature from 28°C to 56°C gives the freeze time of 13.7 seconds (27% increase).



Effects of processing conditions on the freeze time, changing one thing at a time:

- Changing the polymer materials (this refers to the difference in the value of thermal diffusivity at a given temperature) from LDPE to polystyrene having a thermal diffusivity value of $1.0 \times 10^{-7} \text{ m}^2\text{s}^{-1}$ gives the freeze time of 15.2 seconds (41% increase).

What do you predict about cooling curves of following polymers?

- **Semi-crystalline polymers**

- **Amorphous polymers**



Flow Related Properties

Shrinkage, residual stresses and warpage

Shrinkage (S_v) can be volumetrically defined as:

$$\% S_v = \frac{V_1 - V_0}{V_1} \times 100$$

V_0 is the volumes of the moulding

V_1 is the volumes of the mould.



Flow Related Properties

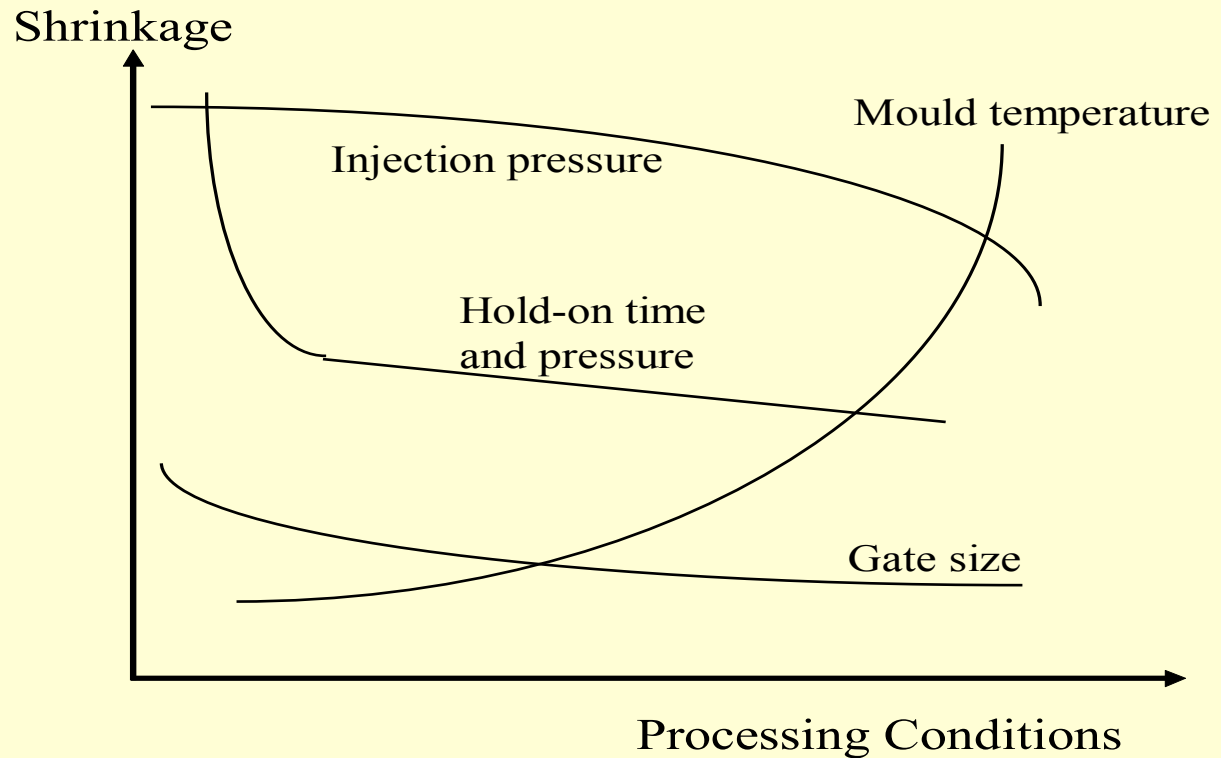
- **General descriptions for shrinkage of polymers**
- **Crystalline materials shrink about 1-4%**
- **Amorphous materials shrink about 0.3-0.7%.**
- **Mouldings with high degree of orientation will shrink more**
- **The percentage shrinkage in the flow direction is usually higher.**
- **Unfilled polymers tend to shrink more.**
- **Generally speaking, processing parameter used to minimise orientation will reduce the shrinkage.**



Effects of Processing conditions on shrinkage

- Mould temperature
- Injection pressure
- Hold-on pressure
- Hold-on time
- Gate size
- Additives

Effects of Processing conditions on shrinkage





Non-uniform shrinkage:

Caused by:

- non-uniform cooling
- geometry of the moulding (having various wall thicknesses of the moulding)
- **Non-uniform shrinkage is also related to:**
- **Internal flow**
- **Warpage**
- **Cold flow**
- **Internal and residual stresses**



Non-uniform shrinkage:

Crystallinity and density

- They are dependent on cooling characteristics.
- In practice, since the cooling rates across the moulding thickness are not uniform the crystalline sizes are expected to be different.
- As the degree of crystallinity rises, the density increases.