Industrial Engineering Laboratory: Plastic Injection Molding

Dr. Chuckaphun Aramphongphun
E-mail: fengchar@ku.ac.th

Plastic Shaping Processes are Important

- Almost unlimited variety of part geometries
- Plastic molding is a *net shape* process
  - Further shaping is not needed
- Less energy is required than for metals due to much lower processing temperatures
  - Handling of product is simplified during production because of lower temperatures
- Painting or plating is usually not required

Two Types of Plastics

1. Thermoplastics
   - Chemical structure remains unchanged during heating and shaping
   - More important commercially, comprising more than 70% of total plastics tonnage
2. Thermosets
   - Undergo a curing process during heating and shaping, causing a permanent change (*cross-linking*) in molecular structure
   - Once cured, they cannot be remelted

Viscosity of Polymer Melts

Fluid property that relates shear stress to shear rate during flow

- Due to its high molecular weight, a polymer melt is a fluid with high viscosity
- Most polymer shaping processes involve flow through small channels or die openings
  - Flow rates are often large, leading to high shear rates and shear stresses, so significant pressures are required to accomplish the processes

Viscosity and Shear Rate

Viscosity of a polymer melt decreases with shear rate, thus the fluid becomes thinner at higher shear rates

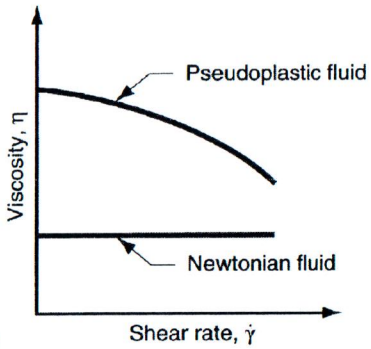


Figure 1. Viscosity relationships for Newtonian fluid and typical polymer melt.

Non-Newtonian Fluids

Viscosity and Temperature

Viscosity decreases with temperature, thus the fluid becomes thinner at higher temperatures

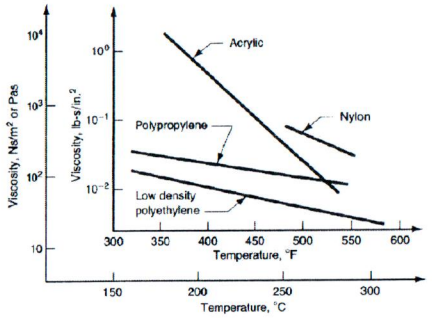


Figure 2. Viscosity as a function of temperature for selected polymers at a shear rate of 10^3 s^{-1}.

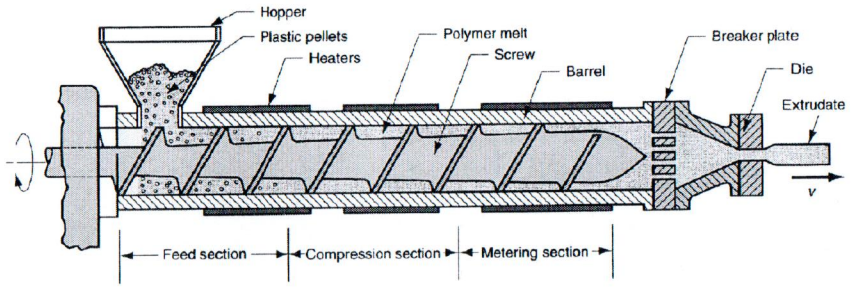
Extrusion

Compression process in which material is forced to flow through a die orifice to provide long continuous product whose cross-sectional shape is determined by the shape of the orifice

- Widely used for thermoplastics and elastomers to mass produce items such as tubing, pipes, hose, structural shapes, sheet and film, continuous filaments, and coated electrical wire
- Carried out as a continuous process; *extrudate* is then cut into desired lengths

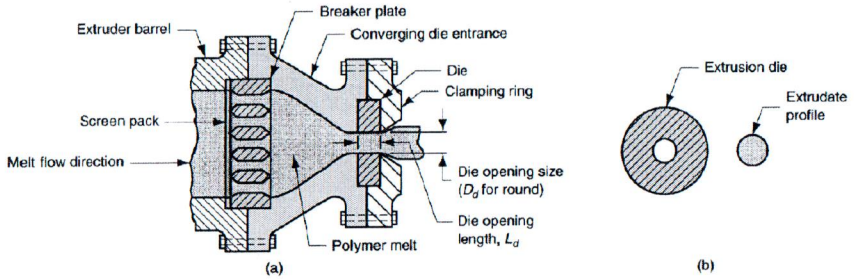
Extruder

Figure 3. Components and features of a (single-screw) extruder for plastics and elastomers



Extrusion Die for Solid Cross Section

Figure 4. (a) Side view cross-section of an extrusion die for solid regular shapes, such as round stock; (b) front view of die, with profile of extrudate. Die swell is evident in both views.



Injection Molding

Polymer is heated to a highly plastic state and forced to flow under high pressure into a mold cavity where it solidifies and the *molding* is then removed from cavity

- Produces discrete components almost always to net shape
- Typical cycle time ~10 to 30 sec, but cycles of one minute or more are not uncommon
- Mold may contain multiple cavities, so multiple moldings are produced each cycle

Injection Molded Parts

- Complex and intricate shapes are possible
- Shape limitations:
  - Capability to fabricate a mold whose cavity is the same geometry as part
  - Shape must allow for part removal from mold
- Part size from ~ 50 g (2 oz) up to ~ 25 kg (more than 50 lb), e.g., automobile bumpers
- Injection molding is economical only for large production quantities due to high cost of mold

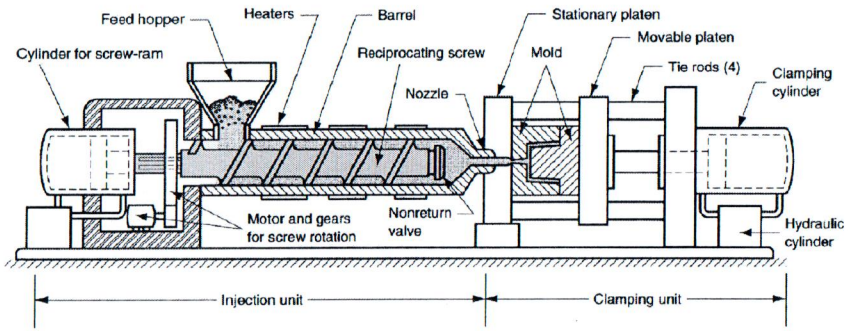
Injection Molding Machine

Two principal components:

1. Injection unit
   - Melts and delivers polymer melt
   - Operates much like an extruder
2. Clamping unit
   - Opens and closes mold each injection cycle

Injection Molding Machine

Figure 5. Diagram of an injection molding machine, reciprocating screw type (some mechanical details are simplified).



Injection Molding Cycle

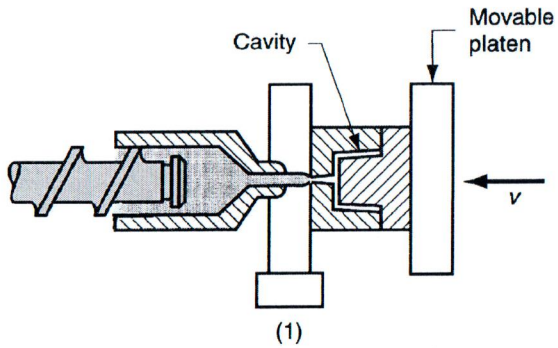


Figure 6. Typical molding cycle: (1) mold is closed.

Injection Molding Cycle

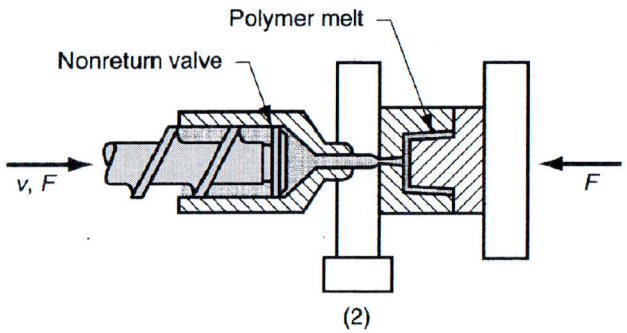


Figure 6. Typical molding cycle: (2) melt is injected into cavity.

Injection Molding Cycle

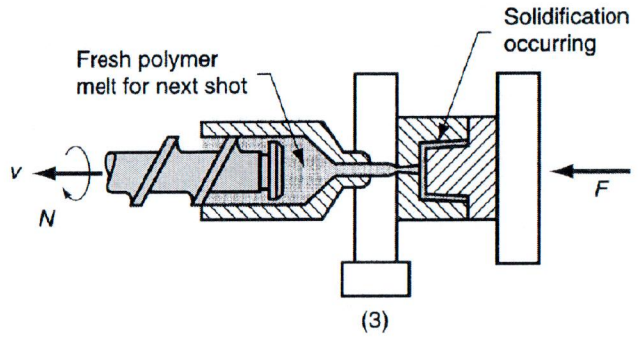


Figure 6. Typical molding cycle: (3) screw is retracted.

Injection Molding Cycle

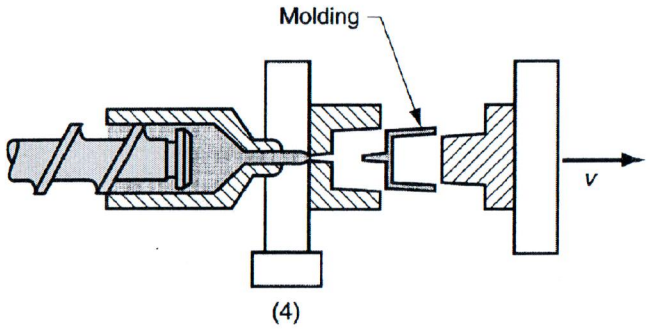


Figure 6.Typical molding cycle: (4) mold opens and part is ejected.

The Mold

- Various types of mold for injection molding:
  - Cold-runner two-plate mold
  - Cold-runner three-plate mold
  - Hot-runner mold

Cold-Runner Two-Plate Mold

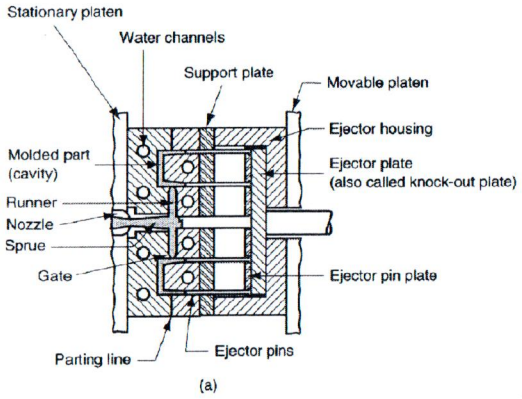


Figure 7. Details of a two-plate mold for thermoplastic injection molding: (a) closed. Mold has two cavities to produce two cup-shaped parts with each injection shot.

Cold-Runner Two-Plate Mold

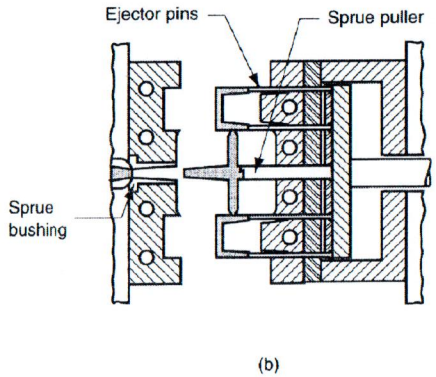


Figure 7. Details of a two-plate mold for thermoplastic injection molding: (b) open

Cold-Runner Two-Plate Mold Features

- Cavity – geometry of part but slightly oversized to allow for shrinkage
  - Created by machining of mating surfaces of two mold halves
- Distribution channel through which polymer melt flows from nozzle into mold cavity
  - Sprue - leads from nozzle into mold
  - Runners - lead from sprue to cavity (or cavities)
  - Gates - constrict flow of plastic into cavity

More Two-Plate Mold Features

- Ejection system – to eject molded part from cavity at end of molding cycle
  - Ejector pins built into moving half of mold usually accomplish this function
- Cooling system - consists of external pump connected to passageways in mold, through which water is circulated to remove heat from the hot plastic
- Air vents – to permit evacuation of air from cavity as polymer melt rushes in

Cold-Runner Three-Plate Mold

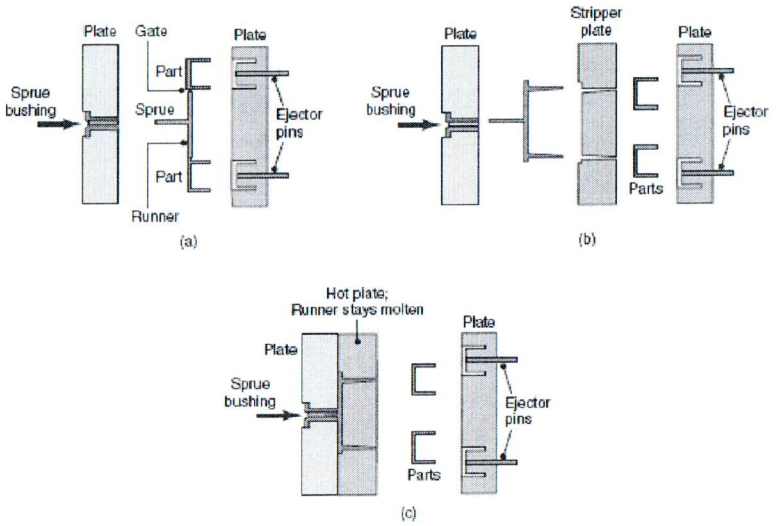
Uses three plates to separate parts from sprue and runner when mold opens

- Advantages over two-plate mold:
  - As mold opens, runner and parts disconnect and drop into two containers under mold
  - Allows automatic operation of molding machine

Hot-Runner Mold

- Eliminates solidification of sprue and runner by locating heaters around the corresponding runner channels
- While plastic in mold cavity solidifies, material in sprue and runner channels remains molten, ready to be injected into cavity in next cycle
- Advantage:
  - Saves material that otherwise would be scrap in the unit operation

Injection Molds



Common Defects in Injection Molded Parts

- Flashing
- Short shot
- Sinks (Sink mark) and Voids

- Moisture splay
- Weld line
- Warpage (Warped parts as ejected from the mold)

Note: Details of these defects in injection molding can be found in the document of injection molding troubleshooting guide.

Lab Assignment

- Observe and operate the injection molding machine to make a plastic part.
- Draw a picture of the molded part and indicate defects found in the part.
- Describe causes and solutions of defects found in the injection molded part according to the document of injection molding troubleshooting guide.
- Summarize 3 possible causes and corrective actions of the defects on an A4 paper sheet with the student's name written on, one sheet / student.
- Keep the plastic part and submit the assignment in the lab assignment box (5^{th} floor, building 8) before 4 PM.