Injection Molding

Outline

- Polymer
- Equipment and process steps
- Considerations for process parameters
- Design for manufacturing, tooling and defects

Materials

Solid materials
- metals
- ceramics
- Plastics
  - thermoplastics
  - thermosets
  - elastomers

Plastics

- $400 billion shipments, 2002 US
- Applications
- Name it

Plastics:
- Music LPs, CDs
- No-sticking TEFLOX
- Stretching SPANDEX
- www.plasticsdatasource.org

Plastic: Greek, plastikos, means to form or mold
Automotive Plastics and Composites Use

- Exterior
  - doors
  - hoods
  - fenders
  - bumper covers (most cars have soft fascia)
- Interior
  - instrument panels, door trim, seats, consoles
- Engine
  - valve covers, intake manifolds, fluid containers, etc.

Plastics, Polymers, Macromolecules

- Poly (many) + mer (structural unit)
  - \([C_2H_4]_n\), poly[ethylene]
- Metal: single atoms, metallic bond
- Ceramic: metallic oxides, ionic bond or dipole interactions, van der Waals bonds

Thermoplastic vs. Thermoset

- Amorphous
- Crystalline
  - (linear)
  - Cross-linked
    - (3D network)

Stiffness
Specific Volume: Amorphous vs. Crystalline

![Graph showing specific volume vs. temperature for amorphous and crystalline materials.](image)

Injection Molding Machine

![Image of an injection molding machine.](image)

Steps of Injection Molding
- Mold closing

![Diagram of mold closing process.](image)

Mold filling

![Diagram of mold filling process.](image)
Packing, holding, cooling

Mold opening, part removal

Injection Cycle Time

- **$$$$**
- **Typical Cycle of Injection Molding**
  - Mold Close: 1-2 sec
  - Injection: 2-5 sec
  - Pack and Hold: 8-10 sec
  - Part Cool: 10-20 sec
  - Screw return: 2-5 sec
  - Mold open: 1 sec
  - Ejection: 1 sec

Injection Molding Cycle
Injection Molding Parameters

- Temperature and Pressure: Function (x,y,z)
- Melt Temperature Control
  - Through Cylinder(Barrel)
    - Frictional Heating
    - Heating bands for 3 zones
      - Rear zone
      - Center zone (10F-20F hotter)
      - Front Zone (10F-20F hotter)
    - Nozzle

Viscosity

- Shear thinning: paints

Pressure Control

- Pressure distribution
- Injection unit
  - Initial injection pressure
    - Applied to the molten plastic and resulting from the main hydraulic pressure pushing against the back end of the injection screw (or plunger).
  - Packing pressure
  - Injection pressure inside mold
    - Usually 1,000 psi to 5,000 psi
    - Lower than hold and pack pressure between 10,000psi and 20,000 psi

Pressure Control

- Hold pressure (packing)
  - Compensate shrinkage
  - Rule of thumb: Hold pressure = 150% of injection pressure.
  - Applied at the end of the initial injection stroke, and is intended to complete the final filling of the mold and hold pressure till gate closure
Pressure Control

Pressure Required

- Total force = projected area times injection pressure (A X P)
- Rule of thumb 4 to 5 tons/in² can be used for most plastics.
- Example,
  - Part is 10 in by 10 in by 1 in
  - Projected area = Surface area = 10 in x 10 in = 100 in²
  - Injection Pressure = 15,000 psi for PC
  - Tonnage required to keep mold closed is
    - \( 100 \text{ in}^2 \times 15,000 \text{ psi} = 1,500,000 \text{ lbs} = 750 \text{ tons} \) (note: 2000 lbs = 1 ton)

Clamp force

Mold cooling

Molding cycle can be shortened by reducing time for cooling and solidification of molten plastics.

Solidification time, \( t \propto \frac{\text{thickness}^2}{\alpha} \), thermal diffusivity

Warpage or stress in a part can be generated when mold shrinkage varies due to different thickness, leading internal residual stress difference.

Even cooling

Mold Temperature Control

- Mold Temperature Control
  - Mold cooling with water, oil.
  - Hot mold for less residual stresses (orientation)
    - Low thermal inertia
  - Uneven cooling
    - Warpage, twisting, shrinkage defects
  - Shrinkage can progress for up to 30 days.
Flow path ratio

Flow path ratio is the ratio between L (the distance between the gate and the farthest point in the molding dimension) and T (the thickness of the part). When molding large or thin parts, the flow path ratio is calculated to determine if molten plastics can fill the mold cavity.

**Rule of thumb**
- Polyethylene (PE) \( L/T = 280-100 \)
- Polypropylene (PP) \( L/T = 280-150 \)
- Polyvinyl chloride (PVC) \( L/T = 280-70 \)
- Polystyrene (PS) \( L/T = 300-220 \)
- Polycarbonate (PC) \( L/T = 160-90 \)
- Acrylonitrile butadiene-styrene (ABS) \( L/T = 280-120 \)
- Polyamide (PA) \( L/T = 320-200 \)

---

Injection molding process window

**Diagram showing the injection molding process window with P<sub>max</sub> and P<sub>min</sub> ranges**

---

Design for Manufacturing

- Part design
  - Moldable
  - Draft angle
  - Shrinkage
  - Reinforcements (ribs and bosses)
  - Cycle time
  - Appearance (defects)
- Mold Design
  - Gate
  - Balancing
- Process Control

---

Draft angle

- For removing parts from the mold
- 1-2°, material, dimension, texture dependent
- Cavity side smaller, core side larger.
- Crystalline material has more shrinkage.
- Amorphous material has smaller shrinkage.
**Shrinkage**

\[ \Delta L = \alpha L \]
\[ \Delta T = \alpha T \]

Resin Shrinkage (%)
- Polyethylene (PE) 1.5-6.0
- Polypropylene (PP) 1.0-3.0
- Polyvinyl chloride (PVC) 0.1-0.5
- Polystyrene (PS) 0.2-0.6
- Polycarbonate (PC) 0.5-0.8
- Acrylonitrile butadienstylene 0.3-0.8
- Polyamide (PA) 0.6-2.0

**Reinforcement**

- **Thickness increase**
- **Stiffer grade material**
  - PP (unfilled), 4,400 psi tensile strength
  - PP (20% glass filled), 7,700 psi
- **Add secondary features, Ribs, bosses**

**Ribs and Bosses**

**Injection Molded Part Design**

- **Base feature + secondary feature (ribs, bosses, holes, etc.)**
- **Nominal wall**: Keep part thickness as thin and uniform as possible.
  - shorten the cycle time, improve dimensional stability, and eliminate surface defects.
  - For greater stiffness, reduce the spacing between ribs, or add more ribs.
- **Nominal wall thickness should be within +/- 10%**
Mold Structure

A dividing line between a cavity plate and a core plate of a mold.
- Make a parting line on a flat or simple-curved surface so that flash cannot be generated.
- Venting gas or air.

Two plate mold

One parting line
Three plate mold

Mold Structure: Undercut, Slide core

Melt Delivery

Sprue
A sprue is a channel through which to transfer molten plastics injected from the injector nozzle into the mold.

Runner
A runner is a channel that guides molten plastics into the cavity of a mold.

Gate
A gate is an entrance through which molten plastics enters the cavity.

Gate
- Delivers the flow of molten plastics.
- Quickly cools and solidifies to avoid backflow after molten plastics has filled up in the cavity.
- Easy cutting from a runner
- Location is important to balance flow and orientation and to avoid defects.
Fan gate, Film gate, Direct gate

Gate Positioning

Point 1: Set a gate position where molten plastics finish filling up in each cavity simultaneously. Same as multiple points gate.
Point 2: Set a gate position to the thickest area of a part. This can avoid sink marks due to molding (part) shrinkage.
Point 3: Set a gate position to an unexposed area of part or where finishing process can be easily done.
Point 4: Consider weldline, molecular orientation.

Runner cross section

Runner cross section that minimizes liquid resistance and temperature reduction when molten plastics flows into the cavity.

- Too big
  - Longer cooling time, more material, cost
- Too small
  - Short shot, sink mark, bad quality
- Too long
  - Pressure drop, waste, cooling

Hot runner, runnerless mold

Runner balancing

Balanced

Not balanced
Defects
Molding defects are caused by related and complicated reasons as follows:

* Malfunctions of molding machine
* Inappropriate molding conditions
* Flaws in product and mold design
* Improper Selection of molding material

Weldline
This is a phenomenon where a thin line is created when different flows of molten plastics in a mold cavity meet and remain undissolved. It is a boundary between flows caused by incomplete dissolution of molten plastics. It often develops around the far edge of the gate.

Cause
Low temperature of the mold causes incomplete dissolution of the molten plastics.

Solution
Increase injection speed and raise the mold temperature. Lower the molten plastics temperature and increase the injection pressure. Change the gate position and the flow of molten plastics. Change the gate position to prevent development of weldline.

Flashes
Flashes develop at the mold parting line or ejector pin installation point. It is a phenomenon where molten polymer smears out and sticks to the gap.

Cause
Poor quality of the mold. The molten polymer has too low viscosity. Injection pressure is too high, or clamping force is too weak.

Solution
Avoiding excessive difference in thickness is most effective. Slow down the injection speed. Apply well-balanced pressure to the mold to get consistent clamping force, or increase the clamping force. Enhance the surface quality of the parting lines, ejector pins and holes.

Short shot
This is the phenomenon where molten plastics does not fill the mold cavity completely, and the portion of parts becomes incomplete shape.

Cause
The shot volume or injection pressure is not sufficient. Injection speed is so slow that the molten plastics becomes solid before it flows to the end of the mold.

Solution
Apply higher injection pressure. Install air vent or degassing device. Change the shape of the mold or gate position for better flow of the plastics.
Warpage

This deformation appears when the part is removed from the mold and pressure is released.

**Cause**
Uneven shrinkage due to the mold temperature difference (surface temperature difference at cavity and core), and the thickness difference in the part. Injection pressure was too low and insufficient packing.

**Solution**
Take a longer cooling time and lower the ejection speed. Adjust the ejector pin position or enlarge the draft angle. Examine the part thickness or dimension. Balance cooling lines. Increase packing pressure.

Sink marks

- Equal cooling from the surface
- Secondary flow
- Collapsed surface

=>$\text{Sink Mark}$

CAE (computer aided engineering)

Process simulation
Material data base
CAD

MOLDFLOW
C-Flow