

2.008

Injection Molding

1

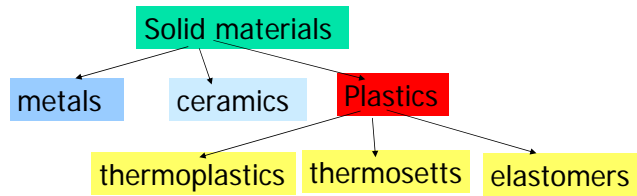
Outline

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

2.008 spring 2004 S. Kim

2

Materials



Plastic: Greek, *plastikos*, means to form or mold

2.008 spring 2004 S. Kim

3

Plastics

- \$400 billion shipments, 2002 US
- Applications Name it
 - Music LPs, CDs
 - No-sticking TEFLON
 - Stre-e-e-tching SPANDEX
 - www.plasticsdatasource.org

2.008 spring 2004 S. Kim

4

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.

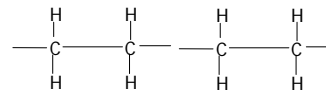


2.008 spring 2004 S. Kim

5

Plastics, Polymers, Macromolecules

- Poly (many) + mer (structural unit)
 $-[C_2H_4]_n-$, poly[ethylene] spaghetti



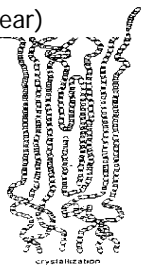
- Metal: single atoms, metallic bond
- Ceramic: metallic oxides, ionic bond or dipole interactions, van der Waals bonds

2.008 spring 2004 S. Kim

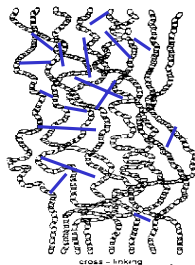
6

Thermoplastic vs. Thermoset

Amorphous
Crystalline
(linear)



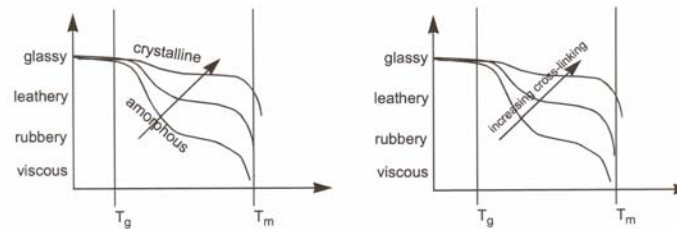
Cross-linked
(3D network)



2.008 spring 2004 S. Kim

7

Stiffness



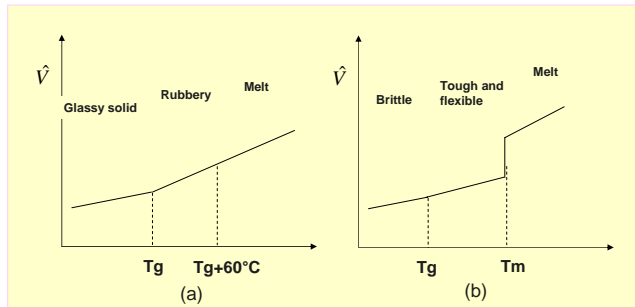
Crystallinity

Cross-linking

2.008 spring 2004 S. Kim

8

Specific Volume: Amorphous vs. Crystalline



2.008 spring 2004 S. Kim

9

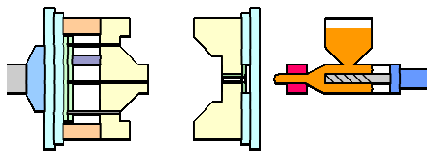
Injection Molding Machine



2.008 spring 2004 S. Kim

10

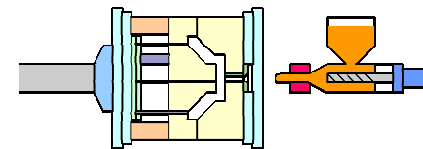
Steps of Injection Molding – Mold closing



2.008 spring 2004 S. Kim

11

Mold filling

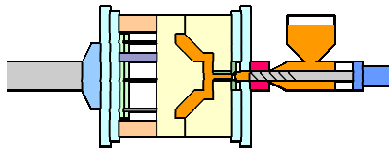


The nozzle will touch the locate ring and then the molten plastic will be injected into the mold.

2.008 spring 2004 S. Kim

12

Packing, holding, cooling

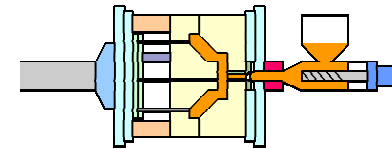


While the molten plastic is being cooled and solidified in the mold, the molding material for the next molding will be plasticized.

2.008 spring 2004 S. Kim

13

Mold opening, part removal



The injection nozzle is separated from the mold, the mold is opened to remove the part.

Ejector pins

2.008 spring 2004 S. Kim

14

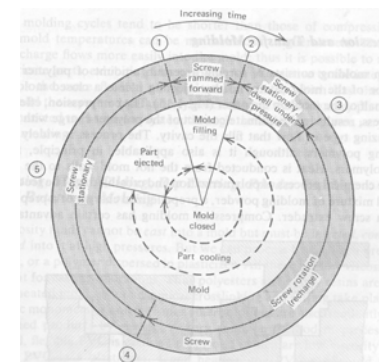
Injection Cycle Time

- \$\$\$
- 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

2.008 spring 2004 S. Kim

15

Injection Molding Cycle



2.008-spring-2004 S. Kim
2.008 spring 2004 S. Kim

Tadmor & Gogos

16

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

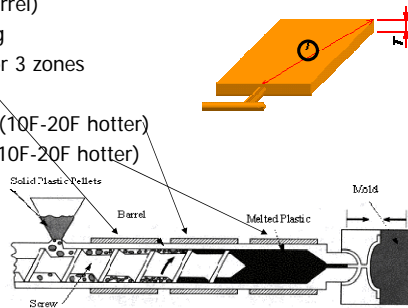


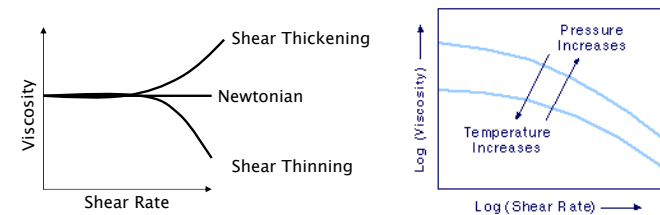
Figure 2: Detail of Screw and Barrel

2.008 spring 2004 S. Kim

17

Viscosity

- Shear thinning: paints

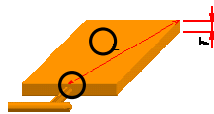


2.008 spring 2004 S. Kim

18

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



2.008 spring 2004 S. Kim

19

Pressure Control

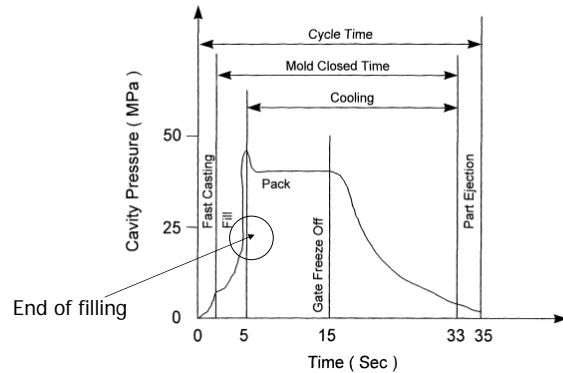
- Hold pressure (packing)
 - Compensate shrinkage
 - Rule of thumb: Hold pressure = 150% of injection pressure.
 - Applied at the end of the initial injection stroke, and is intended to complete the final filling of the mold and hold pressure till gate closure

2.008 spring 2004 S. Kim

20

Pressure Control

PRESSURE HISTORY IN AN INJECTION MOLDED PART

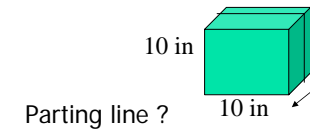


2.008 spring 2004 S. Kim

21

Clamp force

- 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 in² x 15,000 psi = 1,500,000 lbs = 750 tons (note : 2000 lbs = 1 ton)



2.008 spring 2004 S. Kim

22

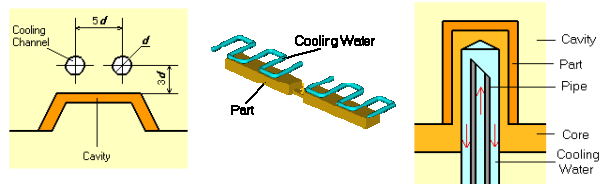
Mold cooling

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

Solidification time, $t \propto \text{thickness}^2 / \alpha$, 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



2.008 spring 2004 S. Kim

23

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.

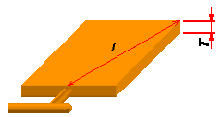
2.008 spring 2004 S. Kim

24

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.



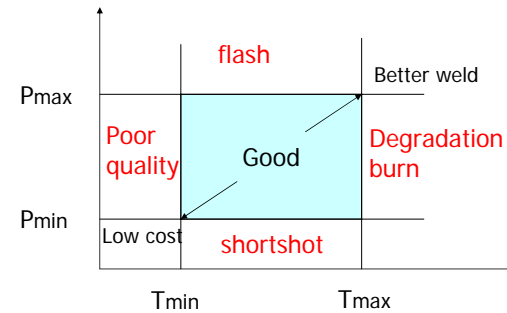
Rule of thumb

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 butadienstyrene (ABS)	L/T = 280-120
Polyamide (PA)	L/T = 320-200

2.008 spring 2004 S. Kim

25

Injection molding process window



2.008 spring 2004 S. Kim

26

Design for Manufacturing

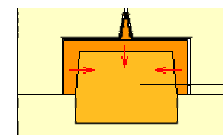
- Part design
 - Moldable
 - Draft angle
 - Shrinkage
 - Reinforcements (ribs and bosses)
 - Cycle time
 - Appearance (defects)
- Mold Design
 - Gate
 - balancing
- Process Control

2.008 spring 2004 S. Kim

27

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.

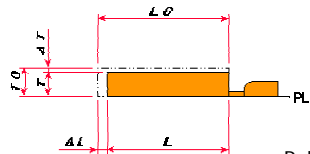


After Cooling Mold

2.008 spring 2004 S. Kim

28

Shrinkage



$$\Delta L = \alpha L$$

$$\Delta T = \alpha T$$

α : shrinkage rate

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 butadienstyrene	0.3-0.8
Polyamide (PA)	0.6-2.0

2.008 spring 2004 S. Kim

29

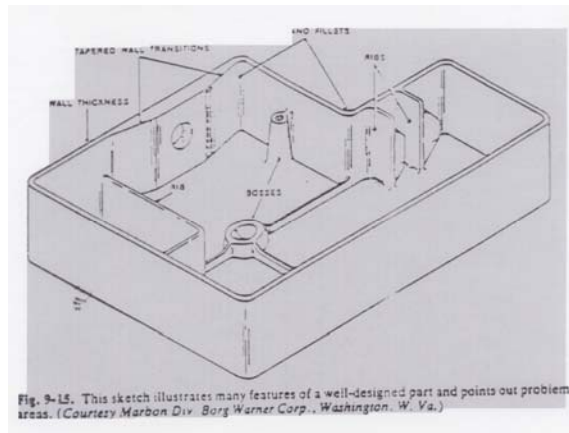
Reinforcement

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

2.008 spring 2004 S. Kim

30

Ribs and Bosses



2.008 spring 2004 S. Kim

31

Injection Molded Part Design

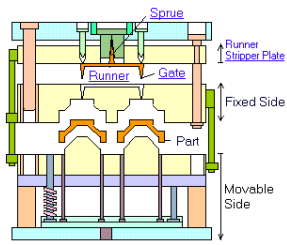
- Base feature + secondary feature (ribs, bosses, holes, etc.)
- Nominal wall : Keep part thickness as **thin and uniform** as possible.
 - shorten the cycle time, improve dimensional stability, and eliminate surface defects.
 - For greater stiffness, reduce the spacing between ribs, or add more ribs.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



2.008 spring 2004 S. Kim

32

Three plate mold

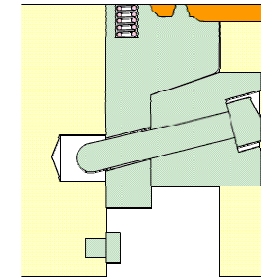
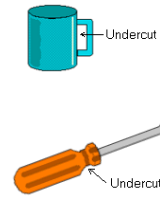


Two parting lines

2.008 spring 2004 S. Kim

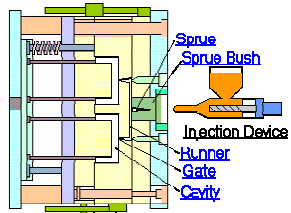
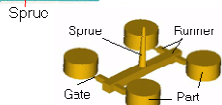
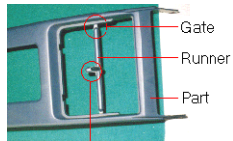
37

Mold Structure: Undercut, Slide core



2.008 spring 2004 S. Kim

38



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

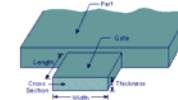
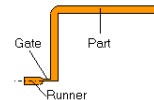
A gate is an entrance through which molten plastics enters the cavity.

2.008 spring 2004 S. Kim

39

Gate

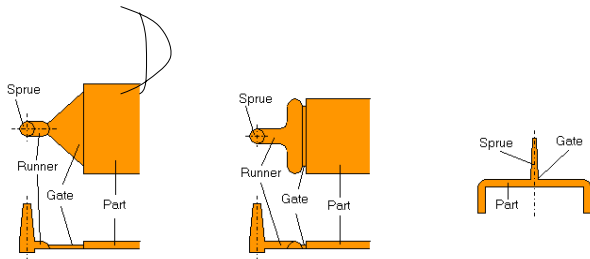
- Delivers the flow of molten plastics.
- Quickly cools and solidifies to avoid backflow after molten plastics has filled up in the cavity.
- Easy cutting from a runner
- Location is important to balance flow and orientation and to avoid defects.



2.008 spring 2004 S. Kim

40

Fan gate, Film gate, Direct gate



2.008 spring 2004 S. Kim

41

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 .

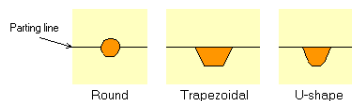
Point 3: Set a gate position to an unexposed area of part or where finishing process can be easily done.

Point 4: Consider weldline, molecular orientation.

2.008 spring 2004 S. Kim

42

Runner cross section



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

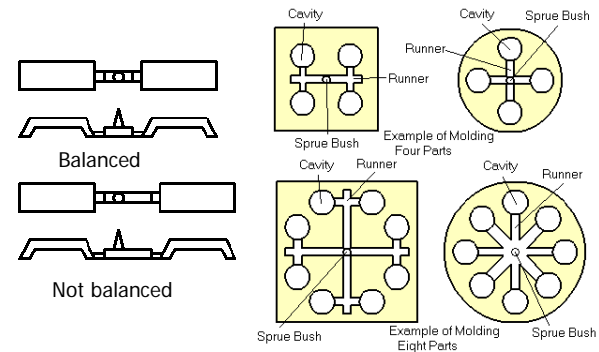
- Too big
 - Longer cooling time, more material, cost
- Too small
 - short shot, sink mark, bad quality
- Too long
 - pressure drop, waste, cooling

Hot runner, runnerless mold

2.008 spring 2004 S. Kim

43

Runner balancing



2.008 spring 2004 S. Kim

44

Defects

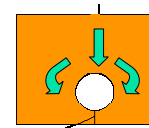
Molding defects are caused by related and complicated reasons as follows:

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

2.008 spring 2004 S. Kim

45

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.

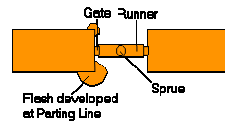
Solution

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.

2.008 spring 2004 S. Kim

46

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.

2.008 spring 2004 S. Kim

47

Short shot

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

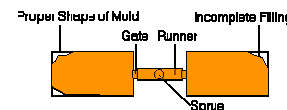
Cause

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

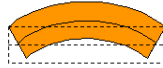
Apply higher injection pressure. Install air vent or degassing device. Change the shape of the mold or gate position for better flow of the plastics.



2.008 spring 2004 S. Kim

48

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.

2.008 spring 2004 S. Kim

49

Sink marks



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

→ Sink Mark

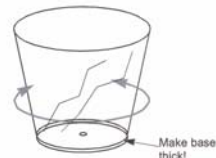
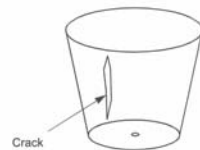
2.008 spring 2004 S. Kim

50

Strength issues



The molecules align in the major direction of flow, and hence there is greater strength.

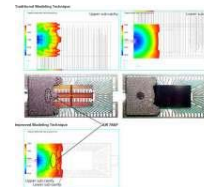


2.008 spring 2004 S. Kim

51

CAE (computer aided engineering)

Process simulation
Material data base
CAD



MOLDFLOW
C-Flow



2.008 spring 2004 S. Kim

52