# 2.008 Design & Manufacturing II

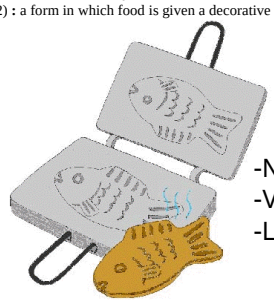
Spring 2004

**Polymer Processing II**
**Injection Molding**



## What is a mold?

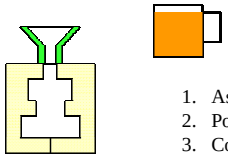
From Webster: a cavity in which a substance is shaped: as (1) **:** a matrix for casting metal (2) **:** a form in which food is given a decorative shape



-Net shape manufacturing
-Volume vs. cost
-Life of a mold



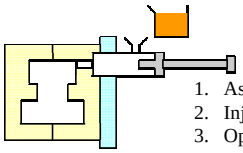
## Casting



1. Assemble a mold.
2. Pour molten metal in.
3. Cool down.
4. Open the mold and remove the part.



## Die casting

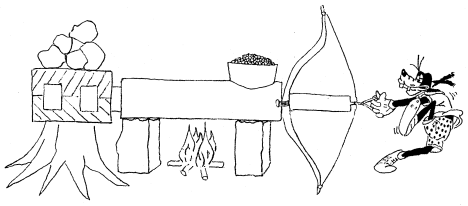


1. Assemble a mold
2. Inject molten metal into
3. Open and remove

aluminum, magnesium
Precision parts

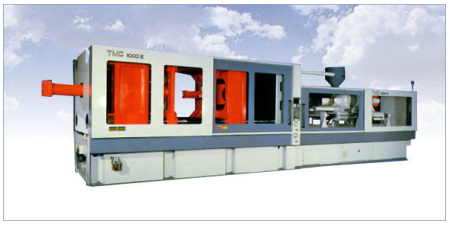


## Injection Molding



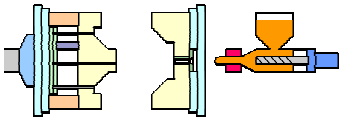


## Injection Molding Machine



# Steps of Injection Molding

-Mold closing
-Filling, packing, holding
-Cooling
-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

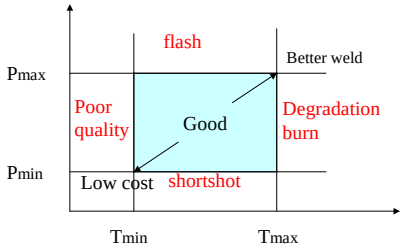


# Design for Manufacturing

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

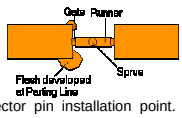


# Injection molding process window






# 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.

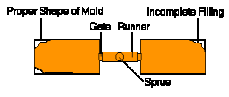
**Cause**

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.

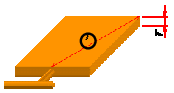
**Solution**

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.



## 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



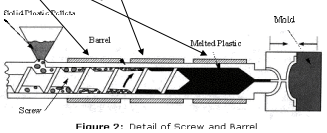
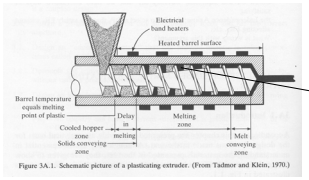
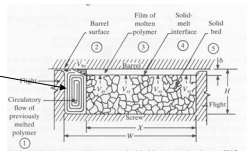


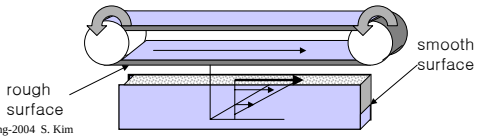

Figure 2: Detail of Screw and Barrel



## Plasticating Extrusion







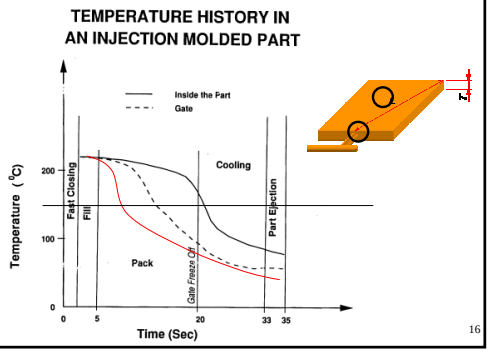



## Suggested Melt Temp at nozzle

| | |
|---|---|
| • Acetal (coplomyer) | 400 F |
| • Acrylic | 425 F |
| • ABS | 400 F |
| • Liquid Crystal Polymer | 500 F |
| • Nylon 6 | 500 F |
| • Polyamide-imide | 650 F |
| • Polyarylate | 700 F |
| • Polycarbonate | 550 F |
| • Polyetheretherketone | 720 F |
| • Polyethylene LDPE | 325 F |
| • Polyethylene HDPE | 350 F |
| • Polypropylene | 350 F |
| • Polystyrene | 350 F |
| • Thermoplastic polyester (PBT) | 425 F |
| • Urethane elastomer | 425 F |



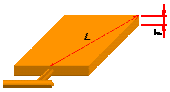
## Temperature Cycle






## 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.



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

Rule of thumb



## 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.

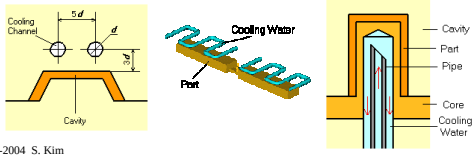
# Mold cooling

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

Solidification time, t ∝ thickness$^2$/$\alpha$, thermal diffusitivity

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

Even cooling






# Warpage

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

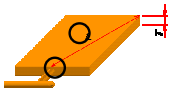
**Cause**
Uneven shrinkage due to the mold temperature difference (surface temperature difference at cavity and core), and the thickness difference in the part. Injection pressure was too low and insufficient packing.

**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.



# Pressure Control

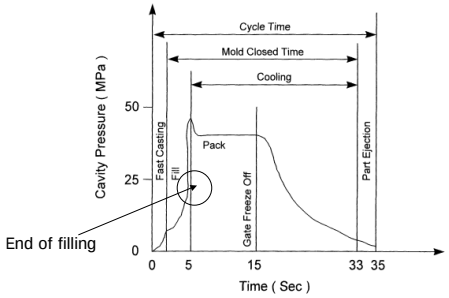
- pressure distribution
- Injection unit
  - Initial injection pressure
    - Applied to the molten plastic and resulting from the main hydraulic pressure pushing against the back end of the injection screw (or plunger).
  - Packing pressure
  - Injection Pressure inside mold
    - Usually 1,000 psi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





# Pressure Control

**PRESSURE HISTORY IN AN INJECTION MOLDED PART**





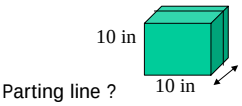

# Pressure Control

– 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

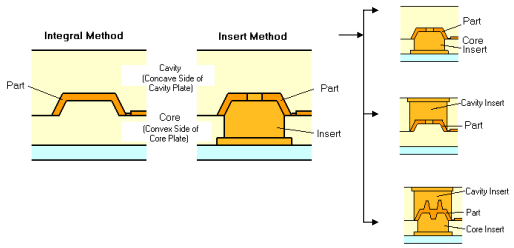


# Clamp force

- Pressure Required
  – Total force = projected area times injection pressure (A X P)
  – Rule of thumb 4 to 5 tons/in$^2$ 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 in$^2$
    - Injection Pressure = 15,000 psi for PC
    - Tonnage required to keep mold closed is
      – 100 in$^2$ x 15,000 psi= 1,500,000 lbs = 750 tons (note : 2000 lbs = 1 ton)



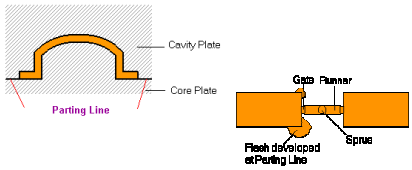
## Mold Structure – Cavity and core





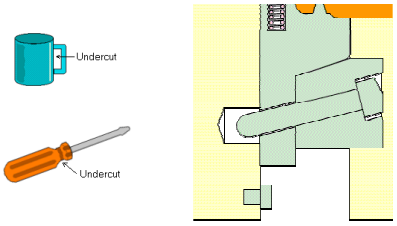

## Mold Structure: Parting line

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.



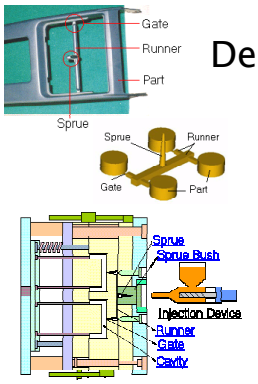



## Mold Structure: Undercut, Slide core






## Delivery



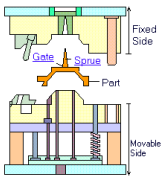

**Sprue**
A sprue is a channel through which to transfer molten plastics injected from the injector nozzle into the mold.

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

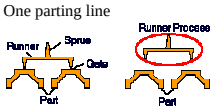
**Gate**
A gate is an entrance through which molten plastics enters the cavity.



## Two plate mold

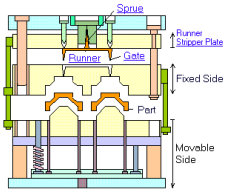



One parting line



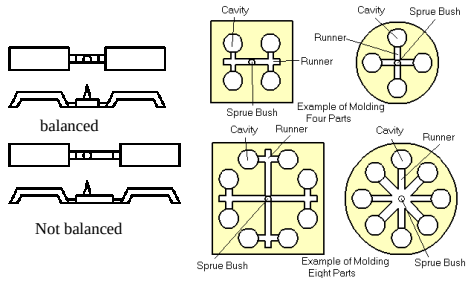



## Three plate mold



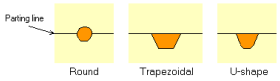

Two parting lines

# Runner balancing






# 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

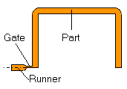


# Gate

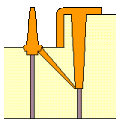
-Restricts the flow and the direction of molten plastics.

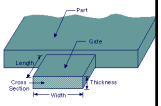
-Quickly cools and solidifies to avoid backflow after molten plastics has filled up in the cavity.

-Simplifies cutting of a runner and moldings to simple finishing of parts.




Side gate

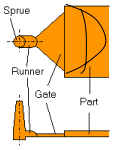


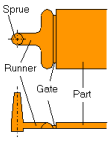



Submarine gate



# Fan gate, Film gate, Direct gate







# Gate Positioning

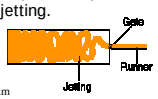
Point 1: Set a gate position where molten plastics finish filling up in each cavity simultaneously. Same as multiple points gate.

Point 2: Basically set a gate position to the thickest area of a part. This can avoid sink marks due to molding (part) shrinkage .

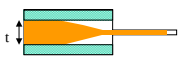
Point 3: Set a gate position to an unexposed area of part or where finishing process can be easily done.

Point 4: Consider degasing, weldline, molecular orientation.

Point5: Fill up molten plastics using the wall surface in order not to generate jetting.

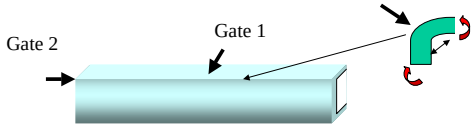


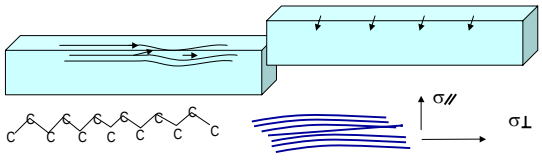

Die swell > Thickness, t






# Molecular orientation



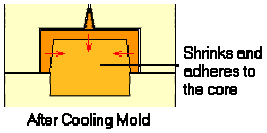
## Design for Manufacturing

- Moldable: flow path ratio, machine size
- Draft angle
- Shrinkage
- Reinforcements (ribs and bosses)
- Cycle time
- Appearance, defects
- Balance, balance, balance!!



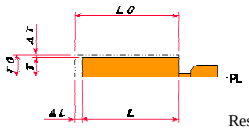
## Draft angle

-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




$\Delta L = \alpha L$
$\Delta T = \alpha T$
$\alpha$: shrinkage rate

| Resin Name | Molding 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 |



## Ribs and Bosses

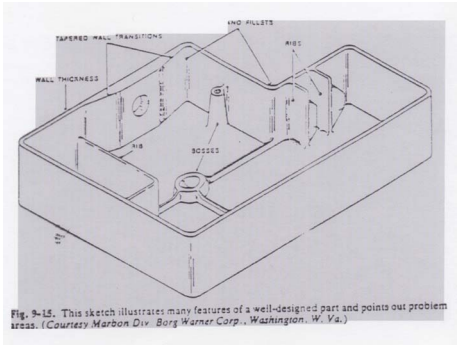
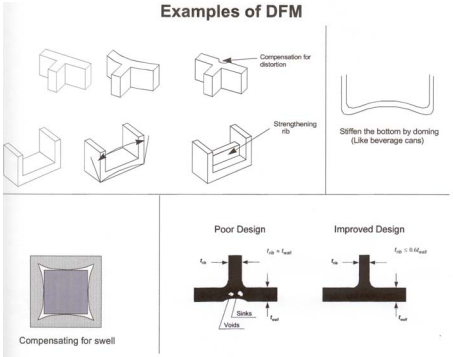



Fig. 9-15. This sketch illustrates many features of a well-designed part and points out problem areas. (Courtesy Marbon Div. Borg Warner Corp., Washington, W. Va.)

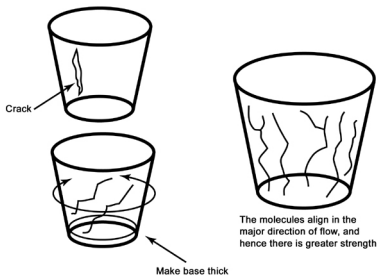


## Examples of DFM






## Strength Issues



# 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



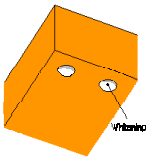
# Sink marks

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

→Sink Mark



# Whitening



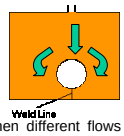

After the ejection by the ejector pin, the surface of the ejected part or surrounding part turns white. When some portion of the part is hard to remove from the mold, that portion also turns white.

**Cause**
The part was hard to remove from the mold. Poor quality of the mold surface

**Solution**
Polish the mold well to facilitate removal of the part. Lower the injection pressure to facilitate removal of the part Reduce the ejector pin speed, and increase the number of ejector pins.



# 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.

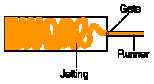


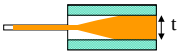
# Jetting

This is the phenomenon where the part has a wire-shape flow pattern on the surface.

**Cause**
Due to inappropriate gate position, a flow of molten plastics into the cavity is cooled in a line shape and remains undissolved with other plastics flow coming later.

**Solution**
Raise the molten plastics and mold temperature, and increase injection speed to make the initial and later flows of molten plastics dissolve completely. Change the gate position to make the molten plastics touch the facing side before making a line shape.




Die swell > Thickness, t



# Flow mark

This is a phenomenon where the initial flow of molten plastics which solidifies mixes with a later flow and remains undissolved. It develops distinctive patterns such as clouds, scales or tree rings.

**Cause**
Injection speed is too fast.
Mold or molten plastics temperature is too low.

**Solution**
Enlarge the gate area to decrease the speed of the molten plastics flowing through the gate.
Increase the pressure retention time for better pressure quality.