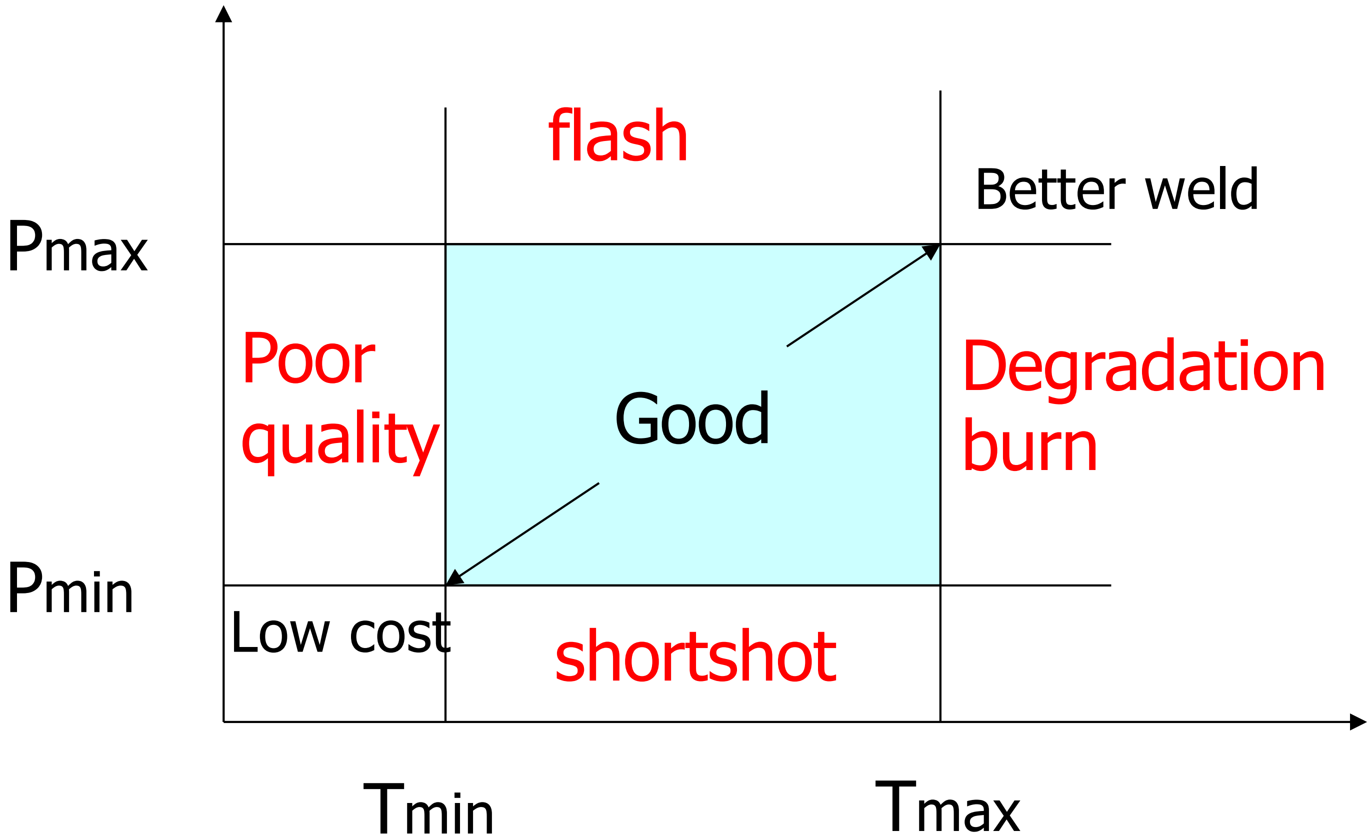
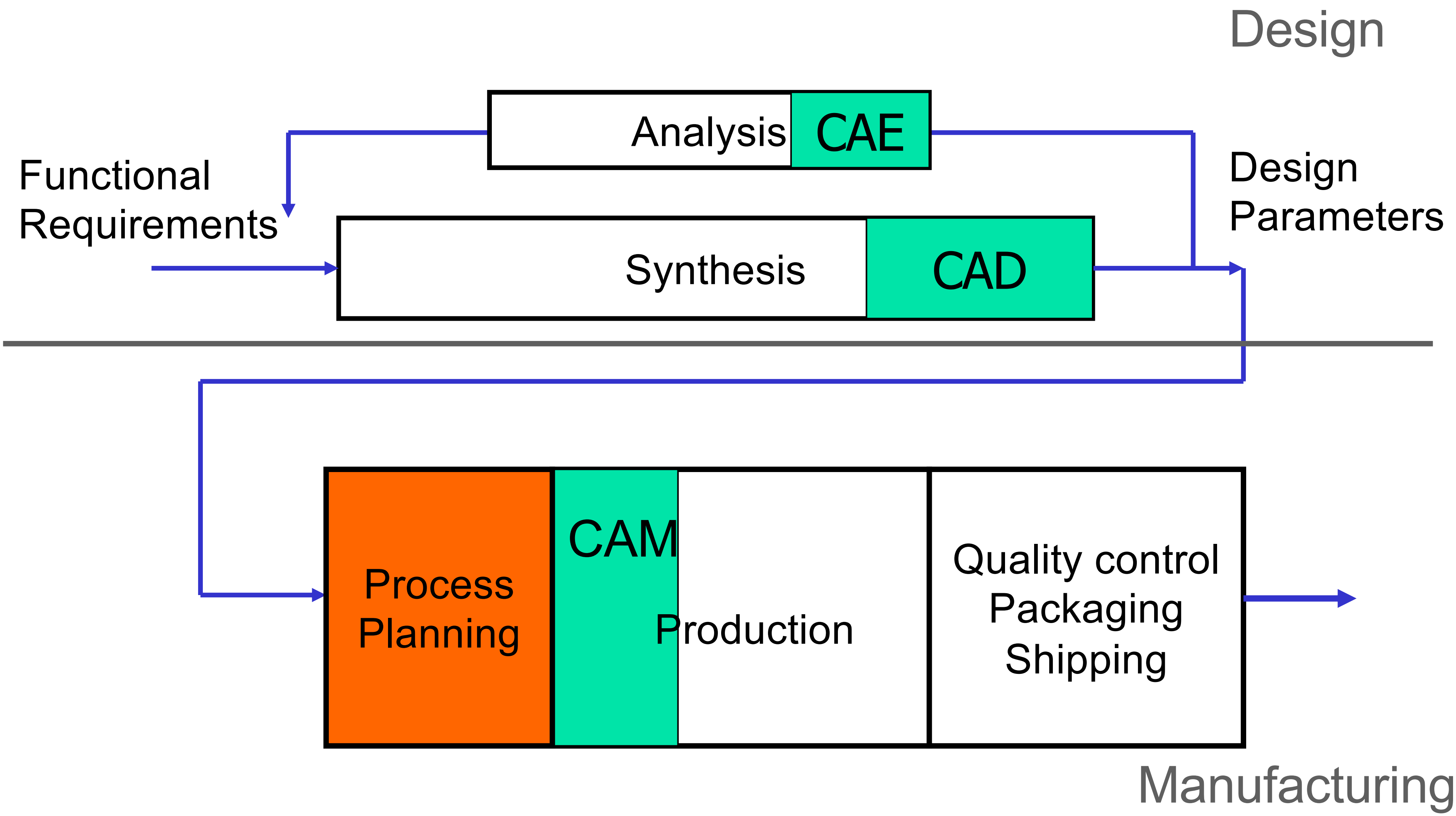
Objectives

- Internalize quality, cost, rate and flexibility as mfg attributes.
- Apply physics to understand the factors that influence the quality, cost, rate and flexibility of the IM process.
- Apply an understanding of variation to the factors that influence the quality, cost, rate and flexibility of processes and systems.
- Understand the impact of manufacturing constraints on product design and process planning.

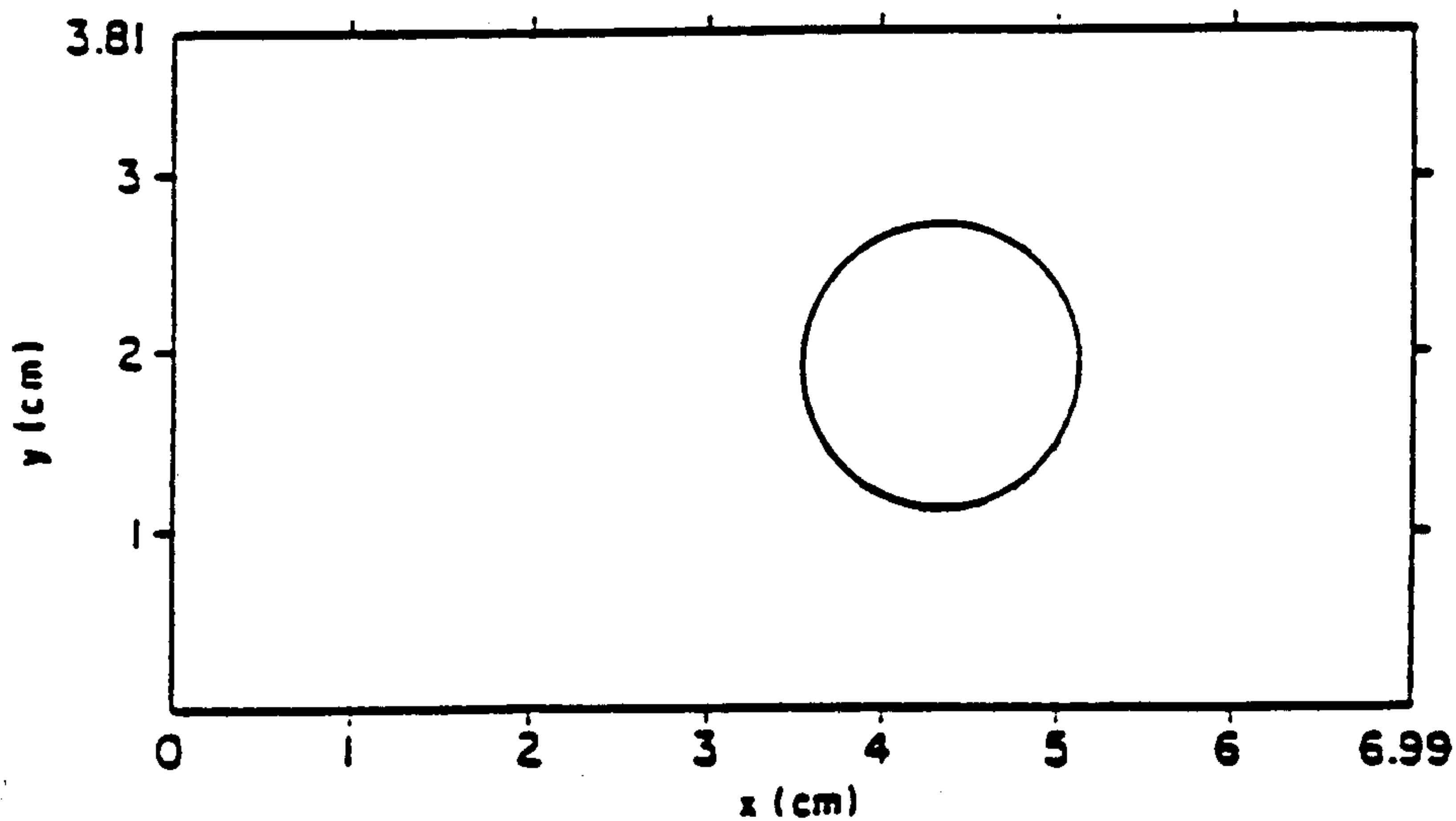
Injection Molding Process Window



Product Development Cycle



Simulated vs. Actual Mold Flow



Simulation: Dotted Line
Actual: Solid Line

2.008

Injection Molding II

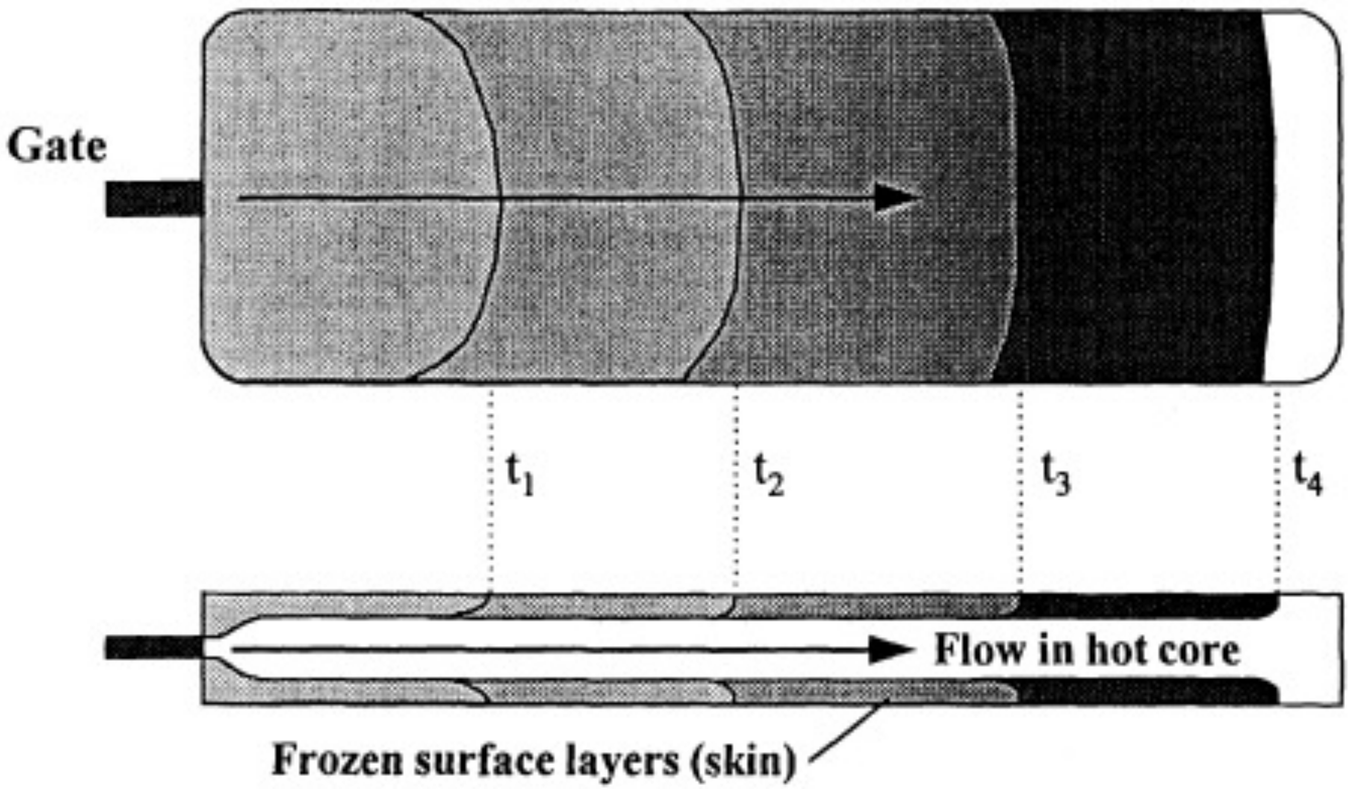
Objectives

- Internalize quality, cost, rate and flexibility as mfg attributes.
- Apply physics to understand the factors that influence the quality, cost, rate and flexibility of processes.
- Apply an understanding of variation to the factors that influence the quality, cost, rate and flexibility of processes and systems.
- Understand the impact of manufacturing constraints on product design and process planning.

Design for Manufacturing

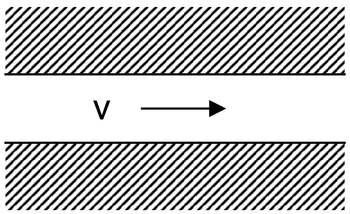
- Part Design
  - Moldable
  - Shrinkage
  - Reinforcements
  - Removable
- Mold Design
  - Mold structure
  - Melt delivery
  - Balancing
- Defects

Non-isothermal Flow



* Source: http://islnotes.cps.msu.edu/trp/inj/flw_froz.html ; ** Z. Tadmore and C. Gogos, "Principles of Polymer Processing"

Non-Isothermal Flow



Flow rate: $1/t \sim V/L$

Heat transfer rate: $1/t \sim a/(T/2)^2$

$$\frac{\text{Flow rate}}{\text{Heat xfer rate}} \sim \frac{V \cdot T^2}{4\alpha \cdot L} = \frac{1}{4}\frac{VT}{\alpha} \cdot \frac{T}{L}$$

Small value
=> Short shot

For injection molding

$$\frac{\text{Flow rate}}{\text{Heat xfer rate}} \sim \frac{1}{4}\frac{10cm/s \times 0.1cm}{10^{-3}cm^2/s} \cdot \frac{0.1cm}{10cm} = 2.5$$

For Die casting of aluminum

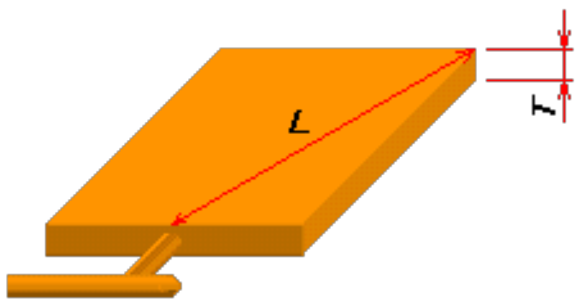
$$\frac{\text{Flow rate}}{\text{Heat xfer rate}} \sim \frac{1}{4}\frac{10cm/s \times 0.1cm}{0.3cm^2/s} \cdot \frac{0.1cm}{10cm} \cong 10^{-2}$$

* Very small, therefore it requires thick runners

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) .

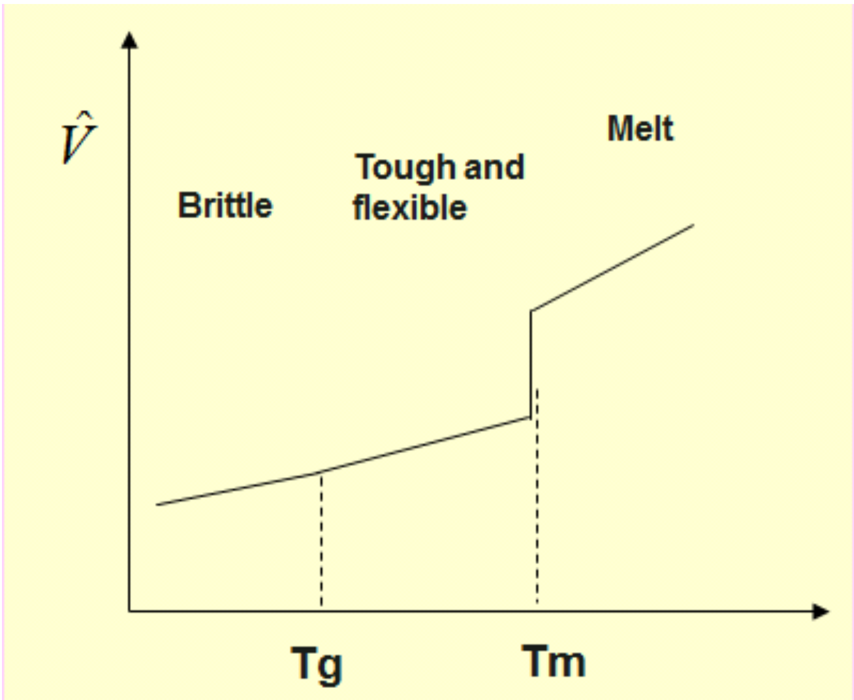
-When molding large or thin parts, the flow path ratio is calculated to determine if molten plastics can fill the mold cavity.



Rule of thumb

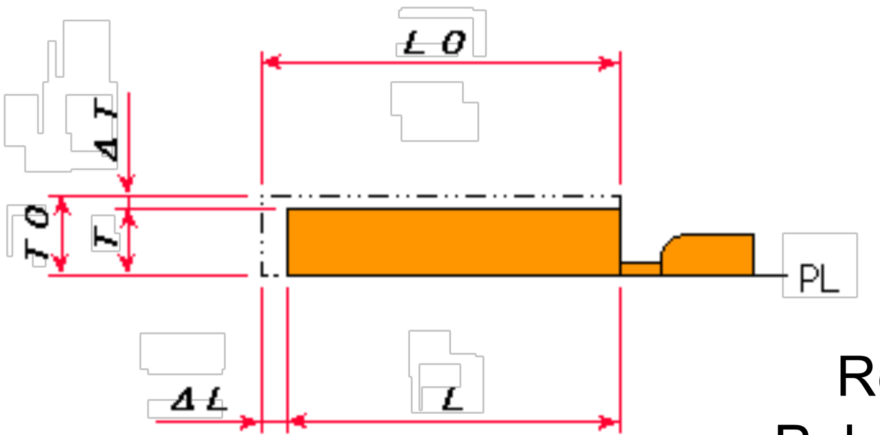
Polyethylene (PE)	L/T = 280-100
Polypropylene (PP)	L/T = 280-150
Polyvinyl chloride (PVC)	L/T = 280-70
Polystyrene (PS)	L/T = 300-220
Polycarbonate (PC)	L/T = 160-90
Acrylonitrile butadienstylene (ABS)	L/T = 280-120
Polyamide (PA)	L/T = 320-200

Specific Volume of Crystalline Thermoplastics



Temperature

Shrinkage



$\Delta L = \alpha L$

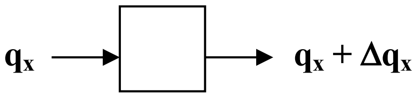
$\Delta T = \alpha T$

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 butadienstylene	0.3-0.8
Polyamide (PA)	0.6-2.0

Heat Transfer Note; $\alpha_{\text{Tool}} \geq \alpha_{\text{polymer}}$

1-dimensional heat conduction equation:



$$\frac{\partial}{\partial t}(\rho \cdot c_p \cdot T)\Delta x \Delta y = -\frac{\partial q_x}{\partial x}\Delta x \Delta y$$

Fourier's law

$$q_x = -k\frac{\partial T}{\partial x}$$

$$\rho \cdot c_p \frac{\partial T}{\partial t} = k\frac{\partial^2 T}{\partial x^2} \quad \text{or} \quad \frac{\partial T}{\partial t} = \alpha \frac{\partial^2 T}{\partial x^2}$$

Boundary conditions:

1st kind $\quad T(x = x') = \text{constant}$

2nd kind $\quad -k\frac{\partial T}{\partial x}(x = x') = \text{constant}$

3rd kind $\quad -k\frac{\partial T}{\partial x}(x = x') = \bar{h}(T - T_\infty)$

The boundary condition of 1st kind applies to injection molding since the tool is often maintained at a constant temperature

Injection Cycle Time

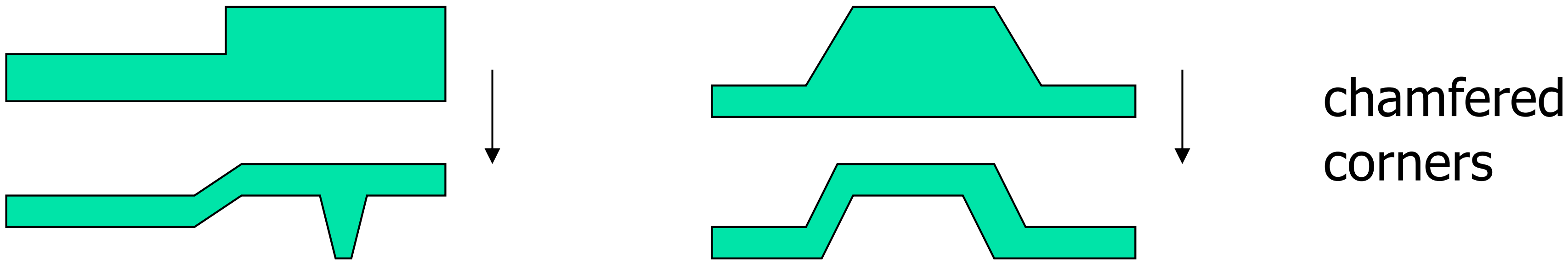
- $$$
- Typical Cycle of Injection Molding
  - Mold Close 1-2 sec
  - Injection 2-5 sec
  - Pack and Hold 8-10 sec
  - Part Cool 10-20 sec
    - Screw return 2-5 sec
  - Mold open 1 sec
  - Ejection 1 sec

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

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%



Ribs and Bosses

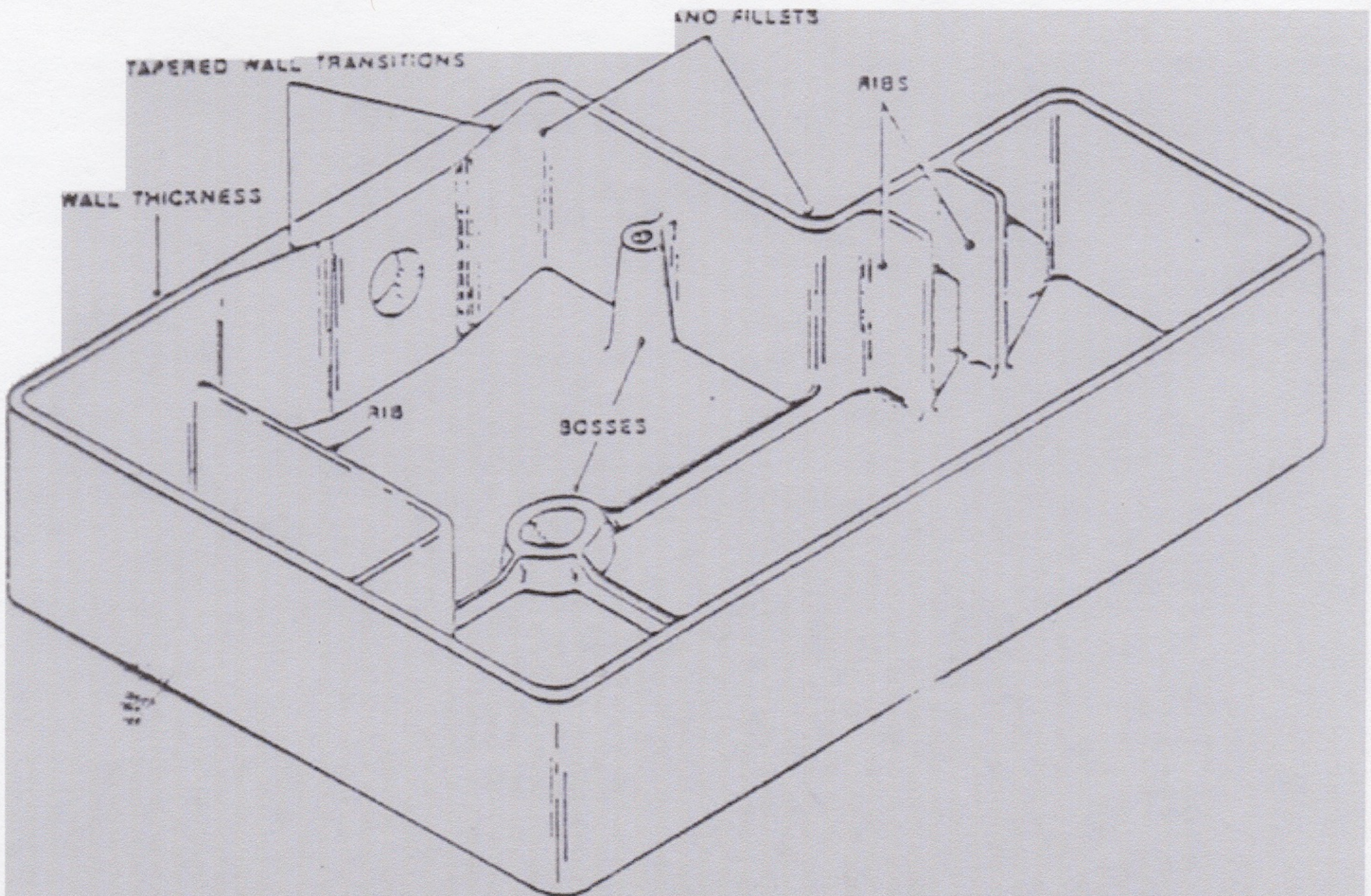
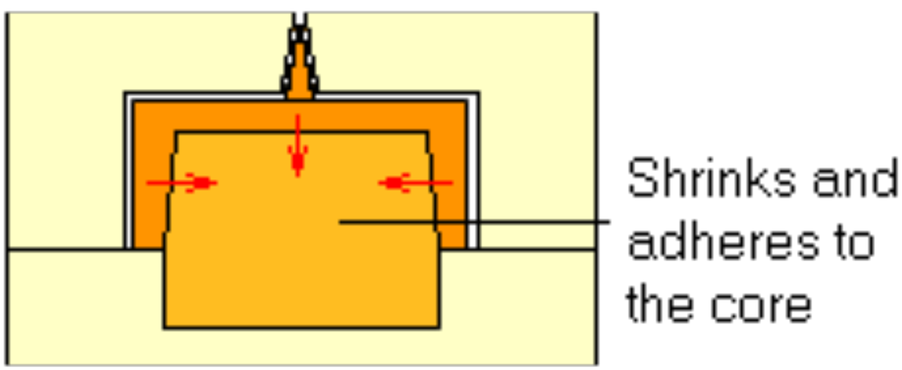


Fig. 9-15. This sketch illustrates many features of a well-designed part and points out problem areas. (*Courtesy Marbon Div Borg Warner Corp., Washington. W. Va.*)

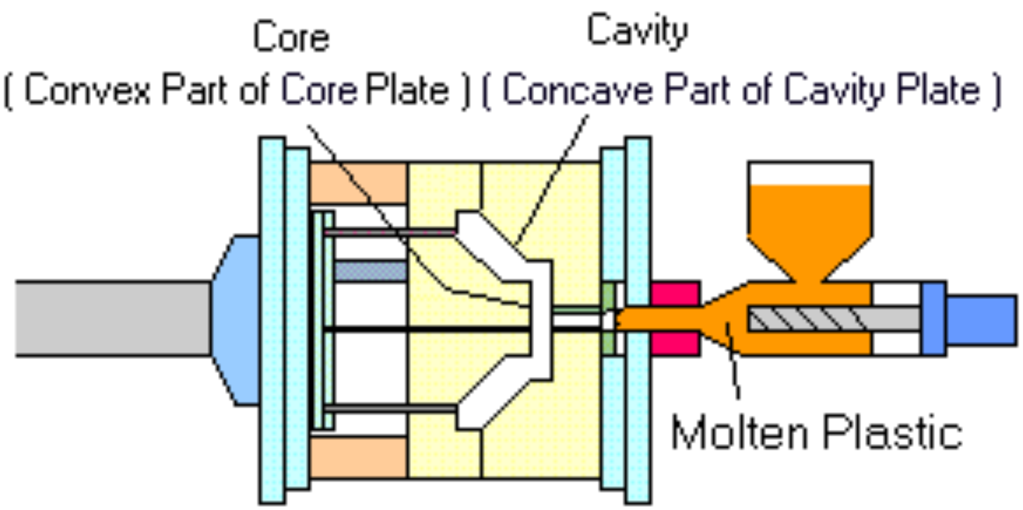
Draft Angle and Parting Line

- For removing parts from the mold
- 1-2^{o}, material, dimension, texture dependent
- Cavity side smaller, core side larger.
- Crystalline material has more shrinkage.
- Amorphous material has smaller shrinkage.

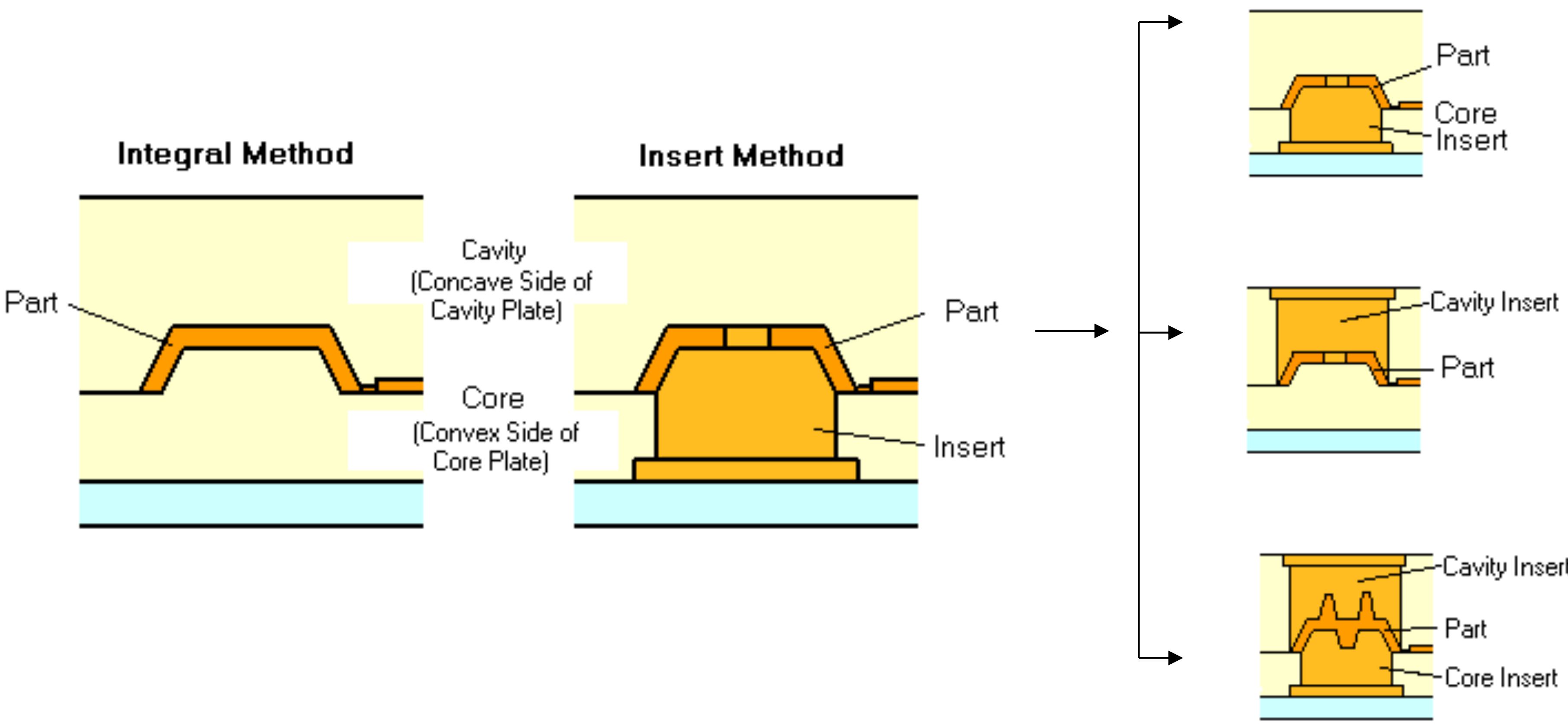


After Cooling Mold

Mold Structure

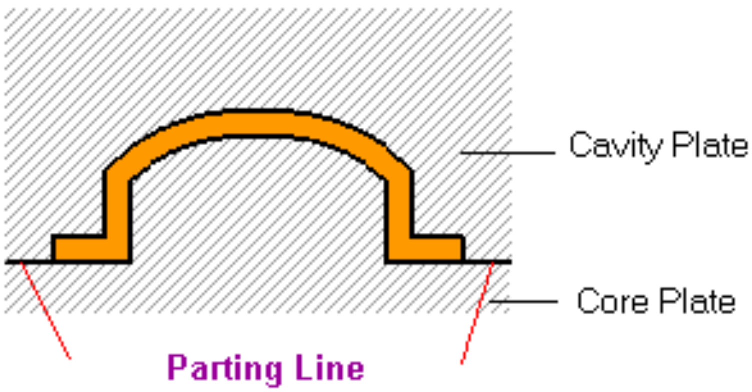


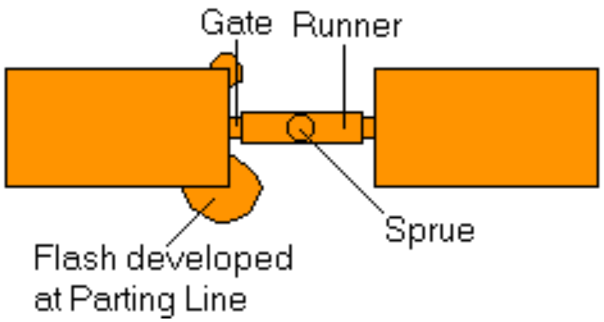
Mold Structure - Cavity and Core



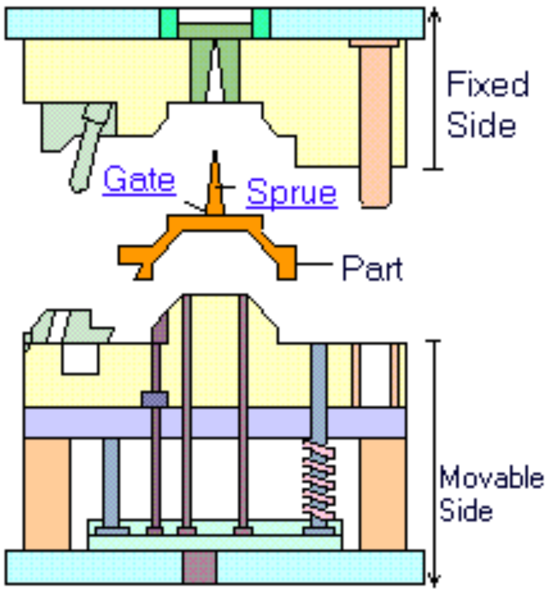
Mold Structure: Parting Line

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.



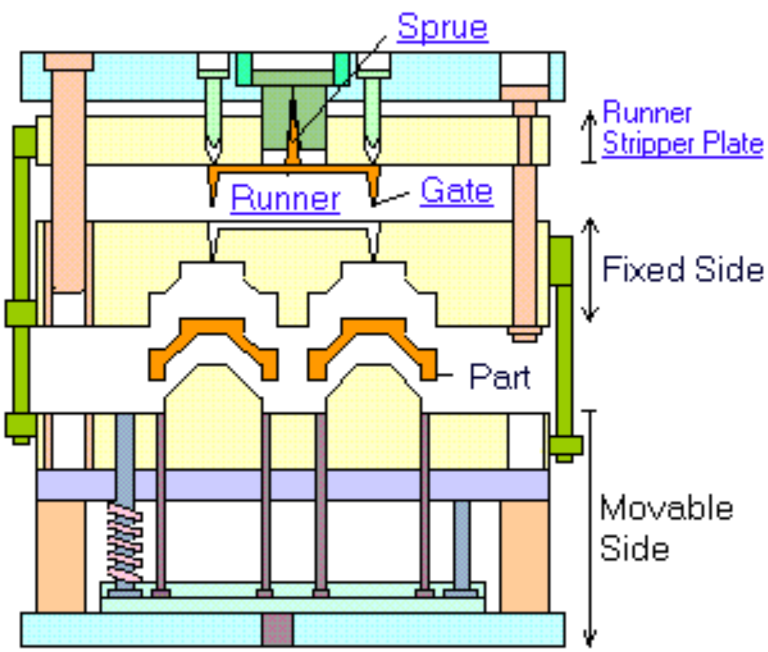


Two Plate Mold



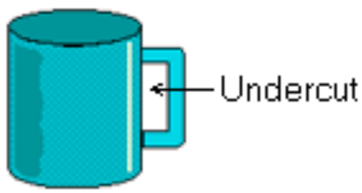
One parting line

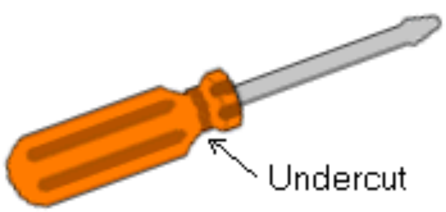
Three Plate Mold

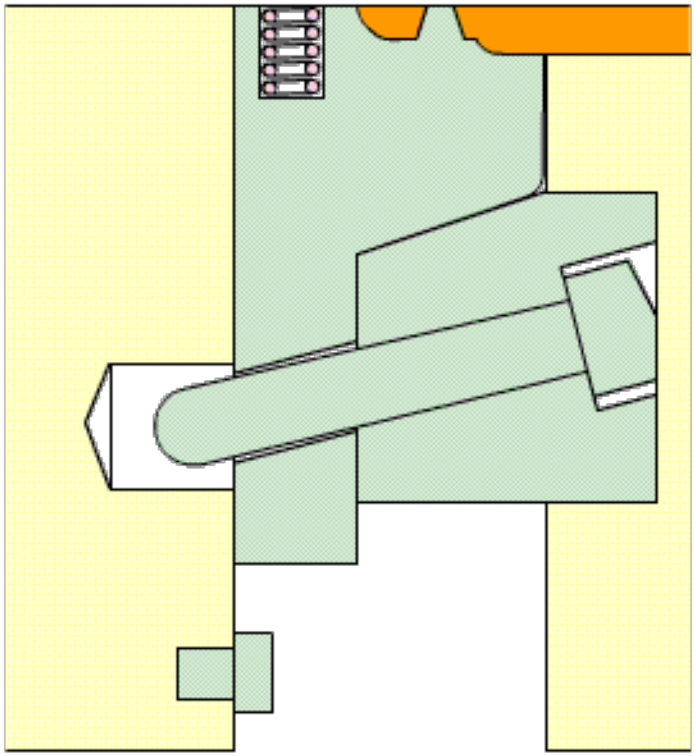


Two parting lines

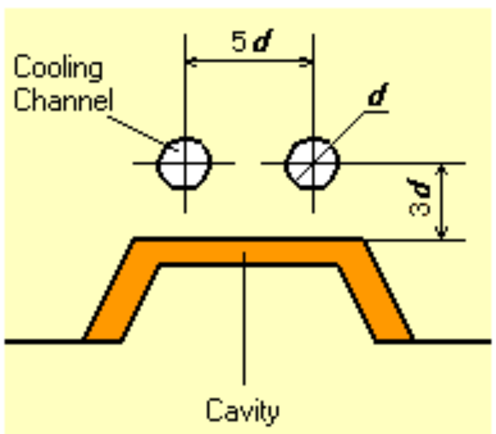
Mold Structure: Undercut, Slide Core

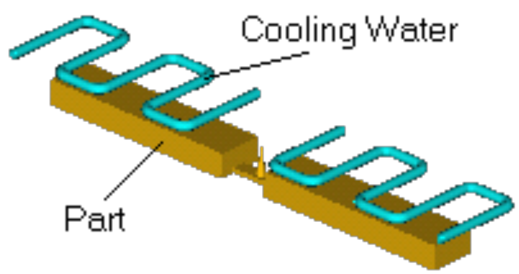


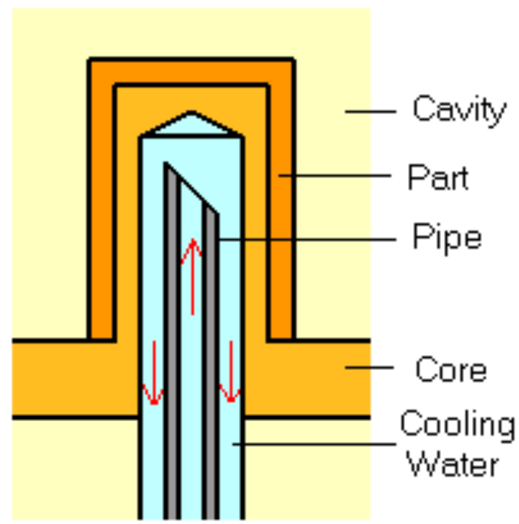


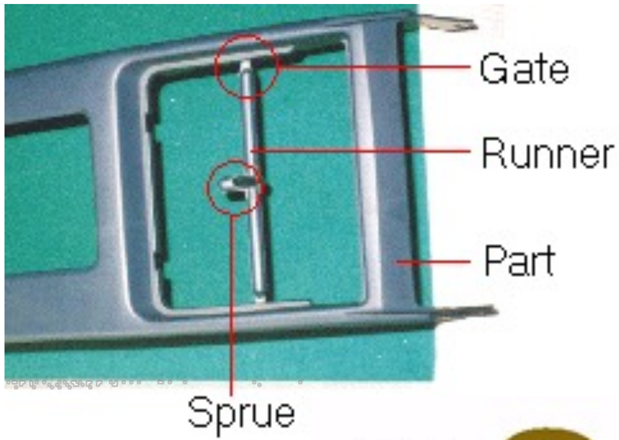


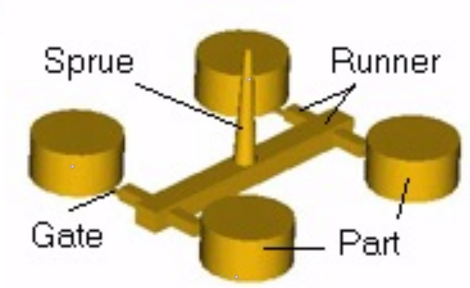
Mold Temperature Control: Even Cooling

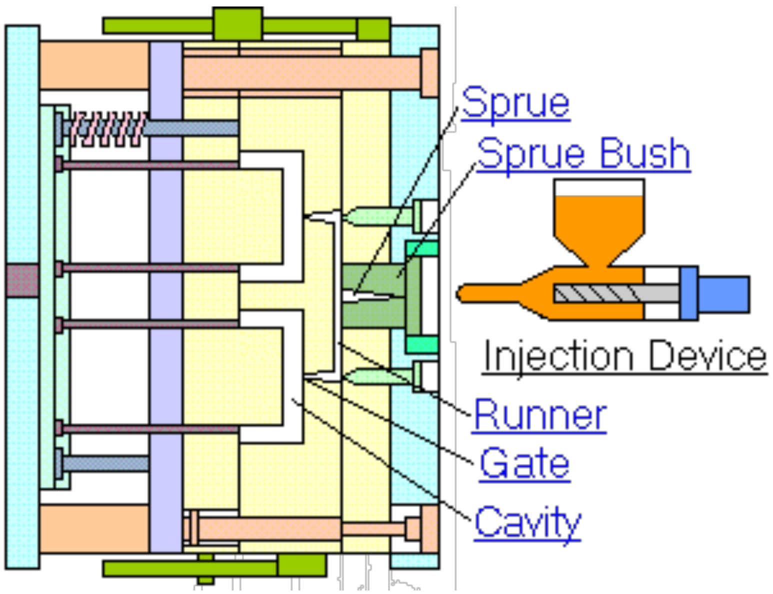












Melt Delivery

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.

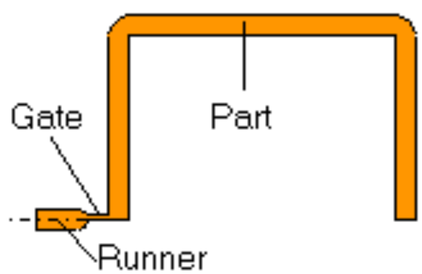
Gate

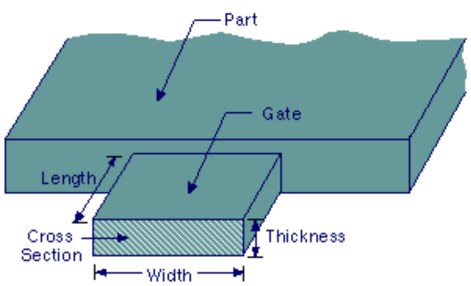
-Delivers the flow of molten plastics.

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

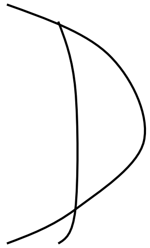
-Easy cutting from a runner

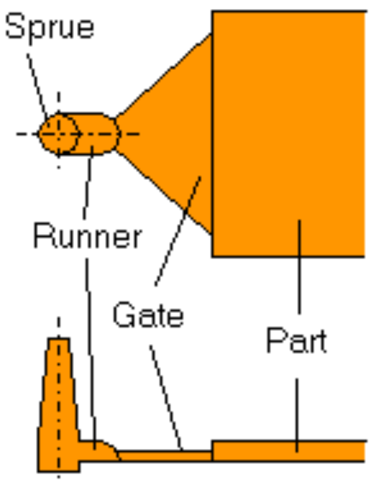
-Location is important to balance flow and orientation and to avoid defects.

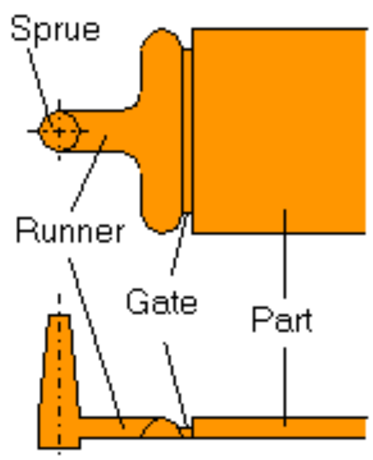


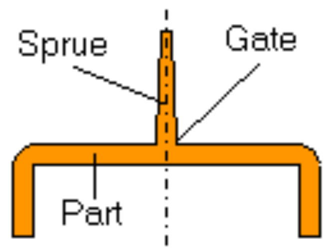


Fan gate, Film gate, Direct gate





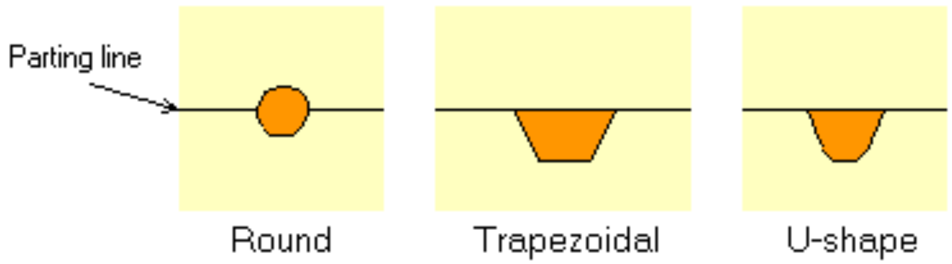




Gate Positioning

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

Runner Cross Section

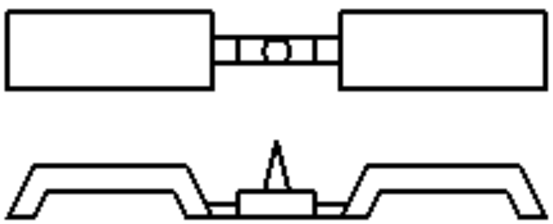


Runner cross section that minimizes liquid resistance and temperature reduction when molten plastics flows into the cavity.

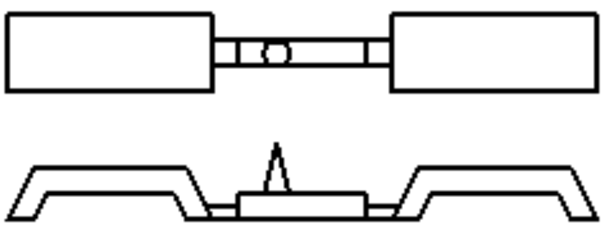
- Too big
    - Longer cooling time, more material, cost
- Too small
    - short shot, sink mark, bad quality
- Too long
    - pressure drop, waste, cooling

Hot runner, runnerless mold

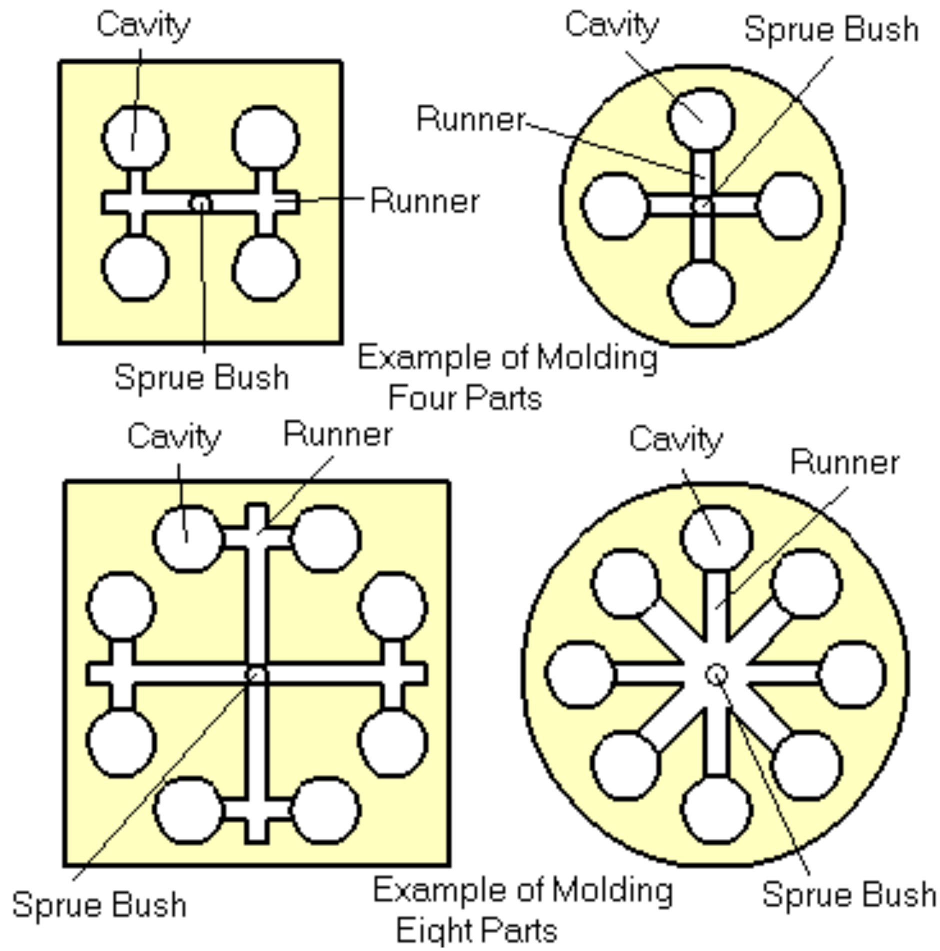
Runner Balancing



Balanced



Not balanced

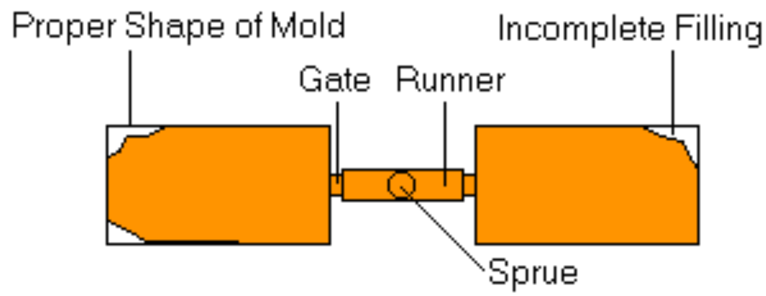


Defects

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

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

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.

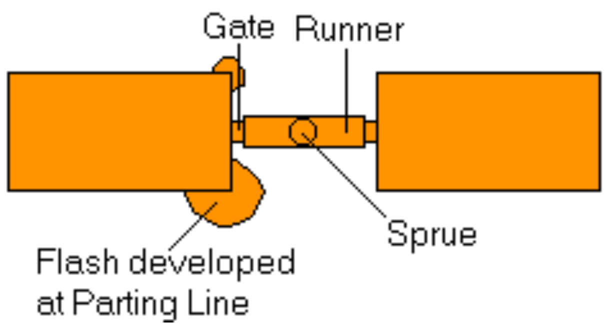
Injection speed is so slow that the molten plastics becomes solid before it flows to the end of the mold.

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.

Short Shots

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.

Injection pressure is too high, or clamping force is too weak.

Solution

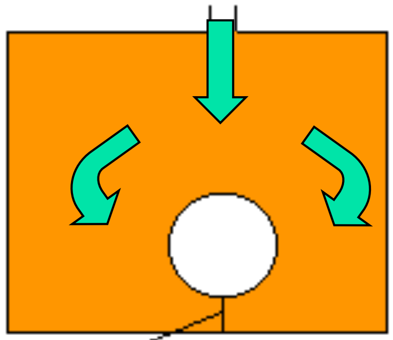
- Slow down the injection speed.
- Apply well-balanced pressure to the mold to get consistent clamping force, or increase the clamping force.
- Enhance the surface quality of the parting lines, ejector pins and holes.

Sink marks

Collapsed surface

Solution: Equal cooling from the surface

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.

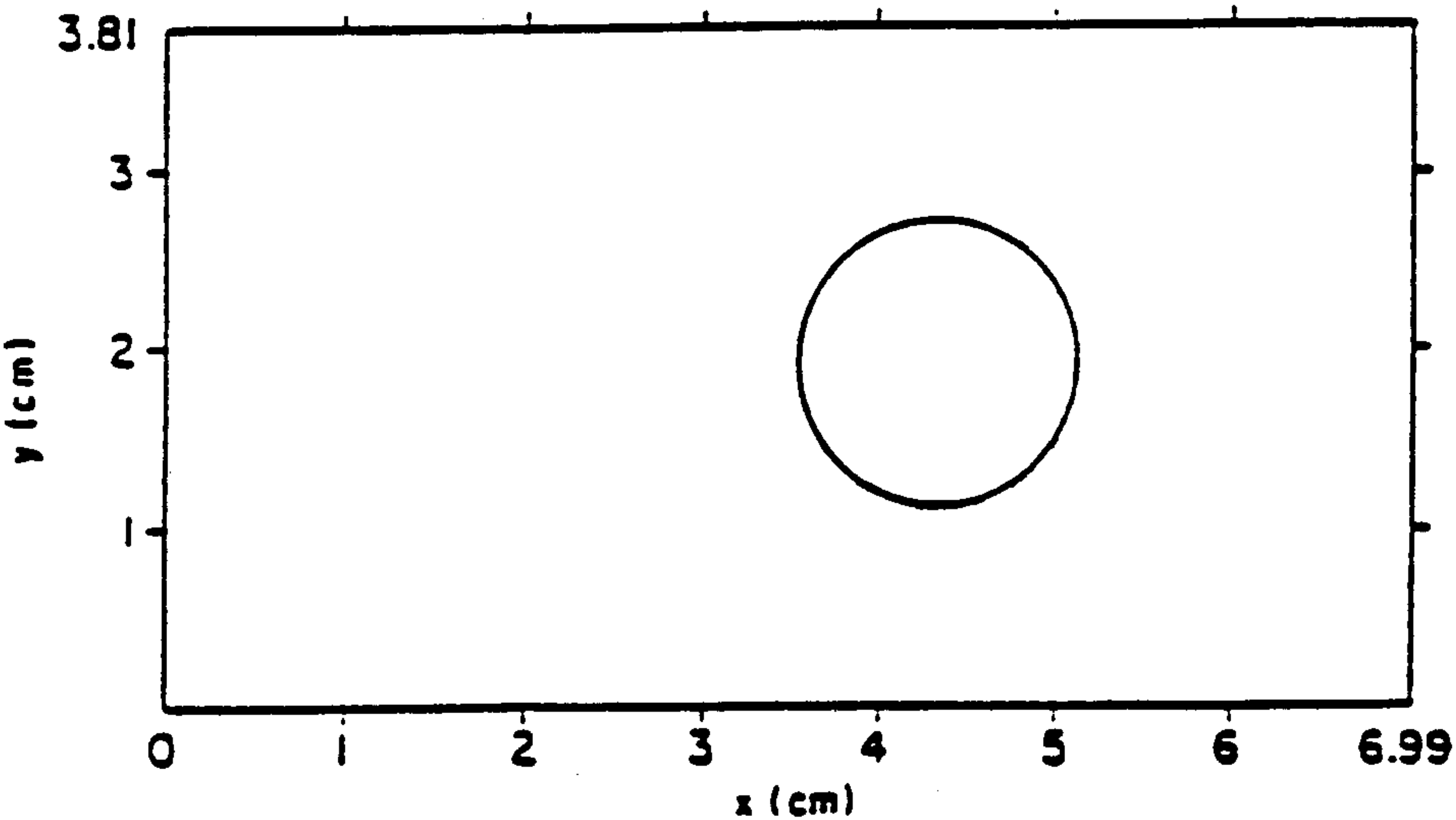
Cause

Incomplete dissolution of the molten plastics.

Solution

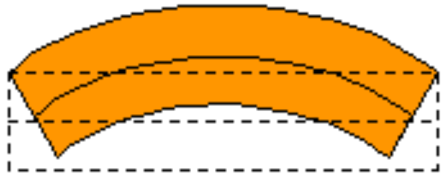
- Increase injection speed and raise the mold temperature.
- Change the gate position and the flow of molten plastics to prevent development of weldline.

Simulated vs. Actual Mold Flow



Simulation: Dotted Line
Actual: Solid Line

Warpage



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

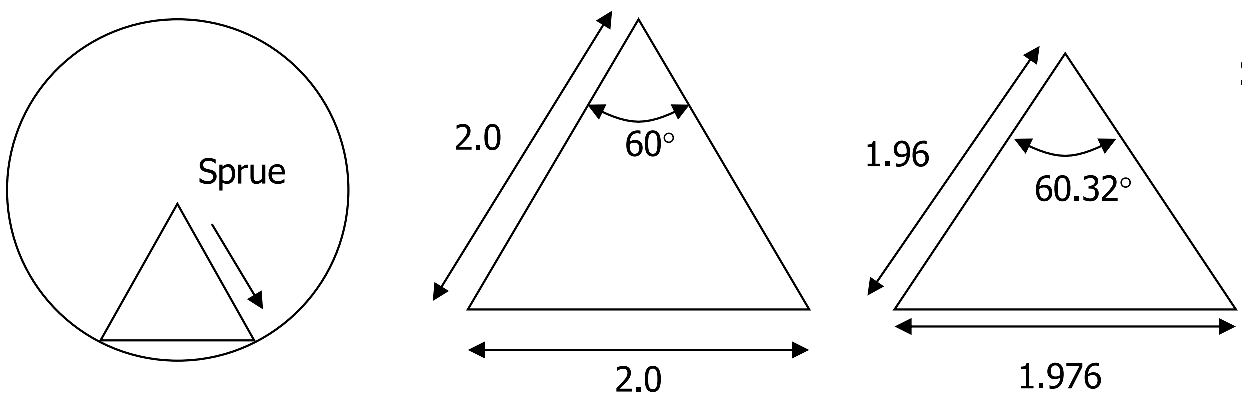
Cause
Uneven shrinkage due to the mold temperature difference (surface temperature difference at cavity and core), and the thickness difference in the part. Injection pressure was too low and insufficient packing.

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.

Mold Temperature Control

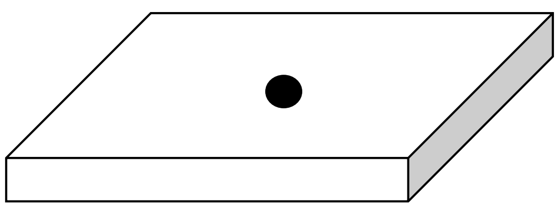
- Hot mold for less residual stresses (orientation)
  - Low thermal inertia
- Even mold cooling with water, oil

Gate Location and Warping

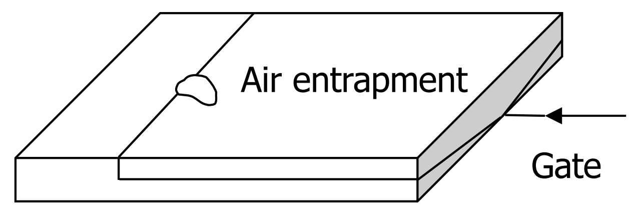


Before shrinkage   After shrinkage

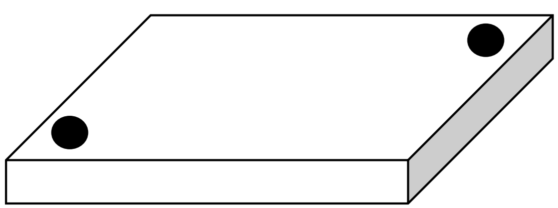
Shrinkage
Direction of flow – 0.020 in/in
Perpendicular to flow – 0.012



Center gate: radial flow – severe distortion



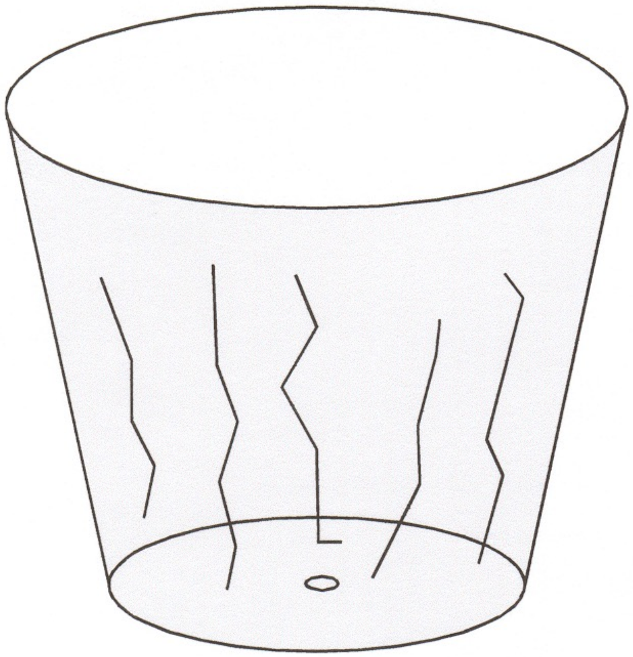
Edge gate: warp free, air entrapment

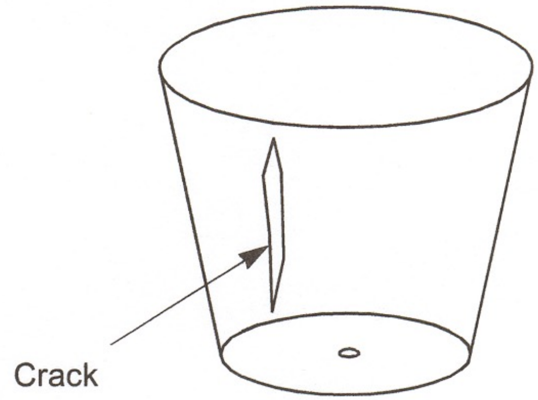


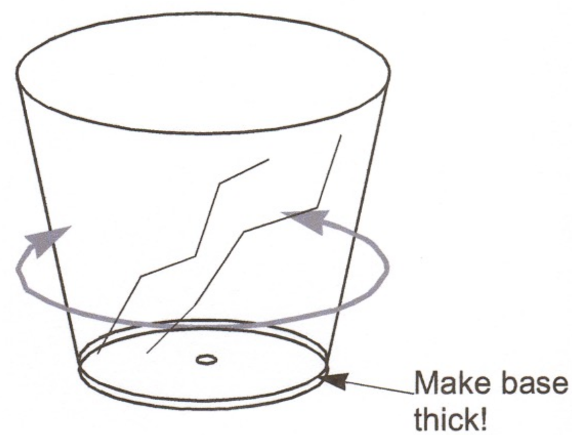
Diagonal gate: radial flow – twisting

End gates: linear flow – minimum warping

Strength issues







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

Image Credits

Slide 8:

Slide 17:

Slides 18-26: