2.008 Design & Manufacturing II

Spring 2004

Polymer Processing III
Thermoforming

Design for Manufacturing

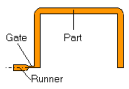
- Moldable: flow path ratio, machine size
- Draft angle
- Shrinkage
- Reinforcements (ribs and bosses)
- Cycle time
- Appearance, defects
- Balance, balance, balance!!

Gate

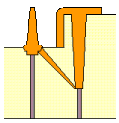
-Restricts the flow and the direction of molten plastics.

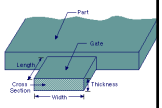
-Quickly cools and solidifies to avoid backflow after molten plastics has filled up in the cavity.

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



Side gate





Submarine gate

Gate Positioning

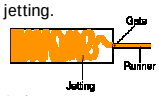
Point 1: Set a gate position where molten plastics finish filling up in each cavity simultaneously. Same as multiple points gate.

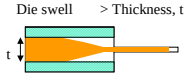
Point 2: Basically set a gate position to the thickest area of a part. This can avoid sink marks due to molding (part) shrinkage .

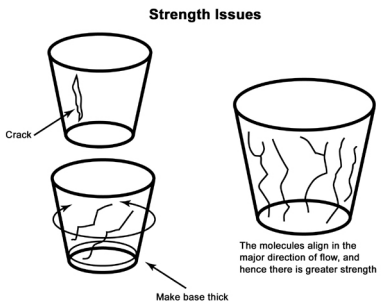
Point 3: Set a gate position to an unexposed area of part or where finishing process can be easily done.

Point 4: Consider degasing, weldline, molecular orientation.

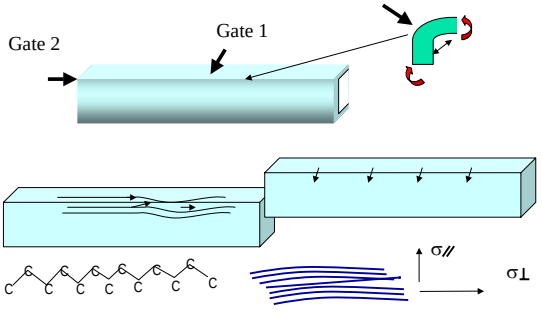
Point5: Fill up molten plastics using the wall surface in order not to generate jetting.







Molecular orientation



Defects

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

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

Sink marks

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

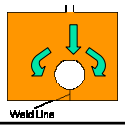
→Sink Mark

Weldline

It is a boundary between flows caused by incomplete fusion 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. Increase the melt temperature and increase the injection pressure. Change the gate position to prevent development of weldline at high stress area.

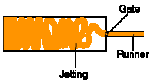


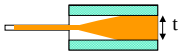
Jetting

This is the phenomenon where the part has a toothpaste 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 unfused with other plastics flow coming later.

Solution
Change the gate position to make the molten plastics touch the facing side before making a line shape.

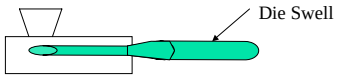




Die swell > Thickness, t

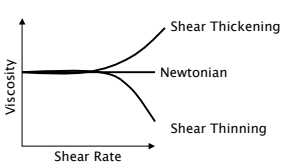
Die swell

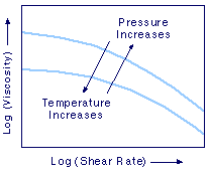
- Exit zone- die
  - die imparts shape on the material, e.g., rod, tube, sheet, channel
  - exit material is called *extrudate*
  - extrudate swells at end of die due to normal forces from the polymer flow, called *die swell*



Viscousity

- Shear thinning: paints





Flow mark

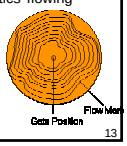
This is a phenomenon where the initial flow of molten plastics which solidifies mixes with a later flow and remains undissolved. It develops distinctive patterns such as clouds, scales or tree rings.

Cause

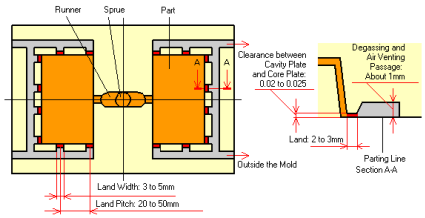
Injection speed is too fast.
Mold or molten plastics temperature is too low.

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.

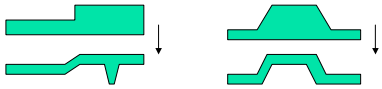


Venting, degassing



Injection Molded Part Design

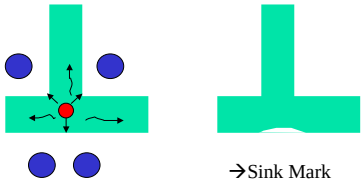
- Base feature + 2ndary feature (ribs, bosses, holes, etc.)
- Nominal wall : Keep part thickness as thin and uniform as possible.
  - shorten the cycle time, improve dimensional stability, and eliminate surface defects.
  - For greater stiffness, reduce the spacing between ribs, or add more ribs.
- Nominal wall thickness should be within +/- 10%



chamfered corners

Uniform cooling

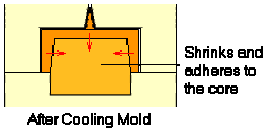
- Differential cooling, differences in shrinkage by different thickness is a leading cause of warpage, sinks, and voids.



→Sink Mark

Draft angle

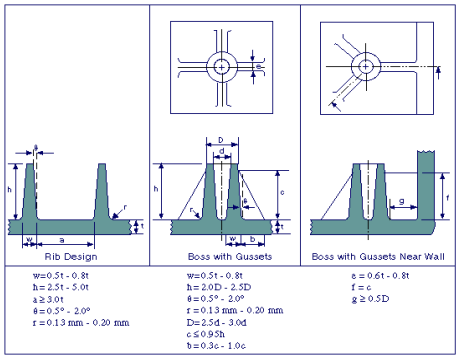
-for removing parts from the mold
- 1-2º, material, dimension, texture dependent
- Cavity side smaller, core side larger.
- Crystalline material has more shrinkage.
- Amorphous material has smaller shrinkage.



Reinforcement

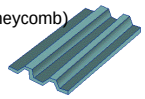
- Thickness increase
- Stiffer grade material
  - PP (unfilled), 4,400 psi tensile strength
  - PP (20% glass filled), 7,700 psi
- Add secondary features, Ribs, bosses

Rib, Boss Design Parameters.



Design rules

- Sink mark, Filling difficulty
- Ribs should be tapered (drafted) at one degree per side.
- The draft will increase the rib thickness from the tip to the root.
- The typical root thickness ranges from 0.5 to 0.8 times the base thickness.
- Ribs aligned in the direction of the mold opening.
- A boss should not be placed next to a parallel wall; instead, offset the boss and use gussets to strengthen it.
- Gussets can be used to support bosses that are away from the walls.
- Ribs can take the shape of corrugations. (honeycomb)



Injection Molding Costs

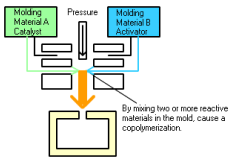
- **Total cost** = Fixed cost + n x Variable cost
- Unit cost = Total cost/n + Variable cost

- Variable Cost
  - Cost of resin and additives
    Additives cost, e.g., colorants, fillers, stabilizers, etc.
  - Material Cost = (resin cost)*(resin fraction) + (additives cost)* (additives fraction)
  - Total Material cost=(part weight +scrap %) x $/lbs
    Scraps from runners, sprues, and part rejects
  - Labor rate=labor cost ($/hr)/(part cycles x #of cavities)
  - **Variable cost**=raw material+labor rate

- **Fixed cost** = Engineering cost+Mold cost +Machine cost+ space

- Engineering cost:
  - Man-hours X $/hr
- Space cost
- Mold costs
  - Type of mold material
  - Machining cost
  - Number of mold sets for the parts needed
- Machine cost
  - Original cost of machine/depreciation time (linear)
  - Special equipment costs for particular jobs, e.g., special controllers or chillers

Reaction Injection Molding (RIM)

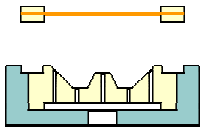


Polyol + Diisocyanate
↓
Polyurethane

Advantages
-As this molding requires lower pressure than regular injection molding, an aluminum or fiber mold can be used. (RRIM)
- Molding large sizes and complicated shapes is possible. (near 100% car bumpers)

Disadvantages
-A copolymerization generates gas, which compresses the air left in the mold and is likely to cause burns.
- Molding cycle is extended.

Vacuum (Thermo) forming



-Soften a sheet of thermoplastics molding material with a heater.
- Suck the air out of the mold through the vent hole to form a vacuum, causing the molding material to conform to the mold and assume its shape.
- Allow air in again to remove the part.

- Sheet
- Heat
- Form
- Cool
- Trim

Advantages

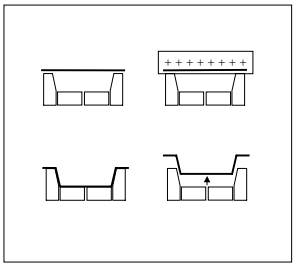
- Low temperature, pressure requirement
- Low mold cost, machine cost
- Large parts
- Fast mold cycles

Disadvantages

- High cost of raw materials (sheets), scraps
- Limited part shapes
- Only one side of part defined by mold
- Inherent wall thickness nonuniformity
- Residual stresses

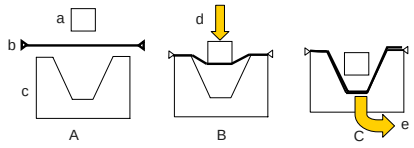
Pressure Forming: Vacuum or Pressure

–Positive air pressure (14.5 to 300 psi)
–faster mold cycles
–lower temperatures with higher forming pressure

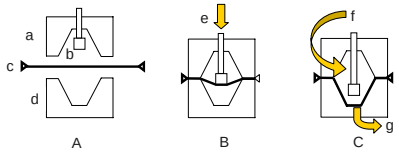


Plug-assisted vacuum forming

–Better wall thickness uniformity especially for cup or box shapes
–Materials of plug include wood, metal, thermoset polymers.
–Plug is 10% - 20 % smaller than cavity.
–Temperature of plug

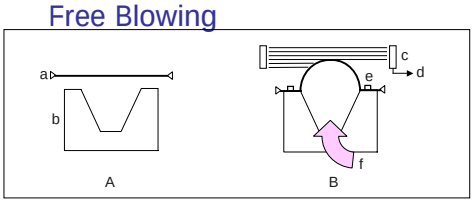


Plug-assisted pressure forming



"a" pressure box "b"plug "c" preheated, clamped sheet "d" female mode with vent holes "e" moving plug "f" applied air pressure "g" venting air

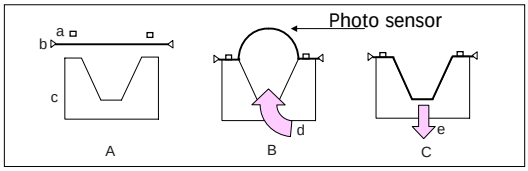
Free Blowing



a - preheated clamped sheet, b - pressure box,
c - proportional photocell monitor, d - signal to air pressure, e - hold-down ring, f - air pressure

–Clamping ring designs can result in controlling shape to hemisphere (circle ring) and elongated (tear drop ring).
–Canopies for racing vehicles.
–Size of bubble not to exceed 50% to 75% of the shorter dimension of clamped sheet.

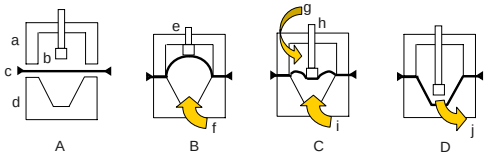
Reverse Draw Forming



a - hold down ring, b - preheated clamped sheet,
c - female mold with pressure/vacuum holes,
d - applied pressure, e - vacuum

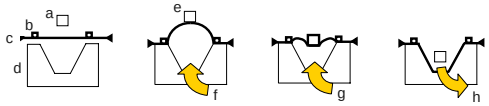
-Better thickness unifromity
-Deep draw
-Longer cycle time

Pressure reverse draw w/ plug assist



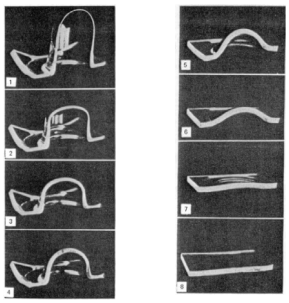
"a" pressure box, "b" plug, "c" preheated, clamped, sheet, "d" female mold with air pressure/vacuum holes, "e" plug begins to move when billow touches it, "f" applied air pressure, "g" air pressure, "h" plug moving into billow, "I" continuing air pressure, "j" vacuum

Vacuum reverse draw w/ plug-assist

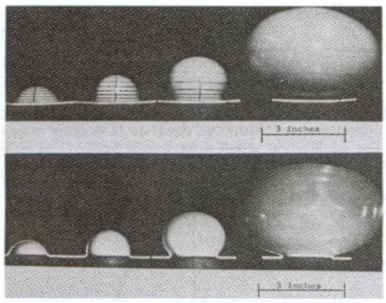


"a" plug "b" hold-down ring "c" preheated, clamped sheet "d" female mold "e" plug motion activated when bubble touches it "f" applied air pressure "g" continuing air pressure as plug advances "h" vacuum

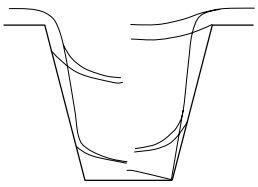
Forming Mechanism



Thickness Uniformity



Forming Considerations: Part Thickness



–Draw ratio = depth of part / width of part.

–Draw ratio should be less than

2:1 for female molds

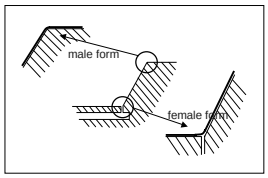
7:1 for male molds

–Area ratio for blank sheet size estimation

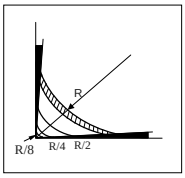
–Draft angle; 2 to 7 degrees

Forming Considerations:

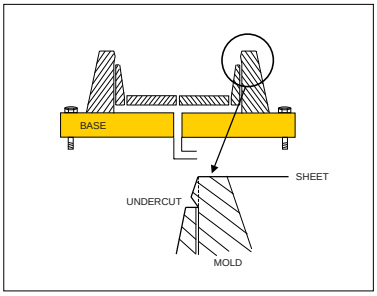
Detail loss



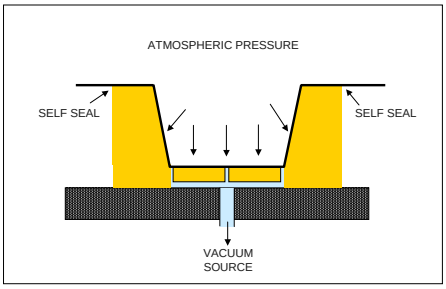
Progressive Draw



Forming Considerations: Undercut



Forming Considerations: Vacuum holes



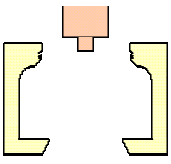
Design for Thermoforming

- Uniform thickness (~10%)
- Simpler shapes (avoid under cuts, etc.)
- Rounded corners (1t min, 4t ideal)
- Draft angle for removal (2 – 7 degree)
- Draw ratio (< 1:1, max 2:1)
- Stretch ratio (< 2:1)
- Shrinkage
- Design for holes and trim lines

Blow Molding

- Packaging, bottles for drinks, containers for cosmetics and toiletries, automotive containers and bumpers.
- Coextrusion products for chemical resistance and structural
- HDPE is the most widely used for high volume packaging
- PP used in processes that promote orientation
- PVC is used for bottles in Europe (homopolymer can be crystal clear) –but temperature, HCl
- PET is primarily used for injection blow molding.

Blow molding



-Pinch a part of a molding material that has been molded into a tube shape with a separate mold.
- Blow compressed air into the molding material, causing it to expand until it conforms to the shape mold to mold the part.

Extrusion blow molding

Extrusion blow molding

- Extrusion Blow Molding the parison is formed from an extrusion die that is similar to one from blown film.
- Extrusion blow molding is discrete. Each part is molded individually.

Injection blow molding

- A parison can have a non-constant cross-section resulting in better wall thickness uniformity than from extrusion blow molding.
- Parisons can be made by injection and then either stored until the finished blow molded parts are needed or shipped to a satellite location where they can be blown.→ shipping cost
- Just oven and a blowing station at the bottling site.

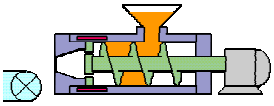
PET bottles

- Performance requirement (after 120 days)
  - less than 15% loss of CO_2
  - no off-taste, no change of shape (swelling), no fall in liquid level
  - drop test of 6 feet with no cracks or leakage, burst test for CO2
  - PET had excellent barrier properties versus PVC (2x), HDPE (52x), PP (57x), and LDPE (114x).
- Stretch blowing development improved properties of PET.
  - PET is injected at 480F-540F and then quenched. (resin is dried)
  - PET preform is heated to 200F (60F higher than that Tg)
  - PET is stretched and blown to form crystals which are small and do not reflect much light.

Blow Film Extrusion

- Products
  - Heavy duty films (0.1 to 0.2 mm) used for covers for agriculture
  - Packaging: wrap, can lining, garbage bags, T-shirt bags, garment
  - Multilayer: (3 to 11 layers) for barrier film
- Process
  - Melting resin in extruder
  - Form molten resin into cylinder or tube.
  - Blow air inside the resin bubble.
  - Pull film into nip rollers through guide rolls.
  - Pull film through a series of rollers.
  - Wind-up film in take-up rolls
  - Bi-axial stretching

Extrusion



- Put a molding material in a hopper (material feed container).
- Plasticate it by stirring and mixing it with a screw while heating it up.
- Push the molding material out by the screw through a small hole of the apex mold (a die used to give the material a desired shape).
- Finish molding by cold solidification.
- Continuous and high productivity
- Constant cross-sections

Advantages & Disadvantages Extrusion

- Advantages
  - Continuous
  - High production volumes
  - Low cost per pound
  - Efficient melting
  - Many types of raw materials
  - Good mixing (compounding)
- Disadvantages
  - Limited complexity of parts
  - Uniform cross-sectional shape only

Thermosets

- Epoxy (bisphenol A + DETA)
  - Excellent chemical and corrosion resistance
  - Excellent thermal properties and low creep
  - High stiffness and adhesion properties
- Polyester(terephthalic acid + ethylene glycol)
  - Rigid, resilient to chemical and environmental exposures, corrosion resistant, and flame retardant
  - Heat or radiation
- Polyurethane (isocyanate and polyol)
  - High strength to weight ratios, resistance to flame spread, excellent thermal insulation, low cost, easily processed

SMC (Sheet Molding Compound)

- SMC is the paste that is compression molded
  - 33% polyester resin and stryrene, which polymerizes and crosslinks
  - 33% glass fibers (1" fibers)
  - 33% Calcium Carbonate

Bulk Molding Compound

- BMC- Resin, fiber, and filler
- Compression Molding

Polyurethane

- Flexible foam, less crosslinking
  - Chemical blowing agent, microcellular
- Rigid urethane, high crosslinking
- Polyurethane can be processed by
  - Casting, painting, foaming
  - Reaction Injection Molding (RIM)