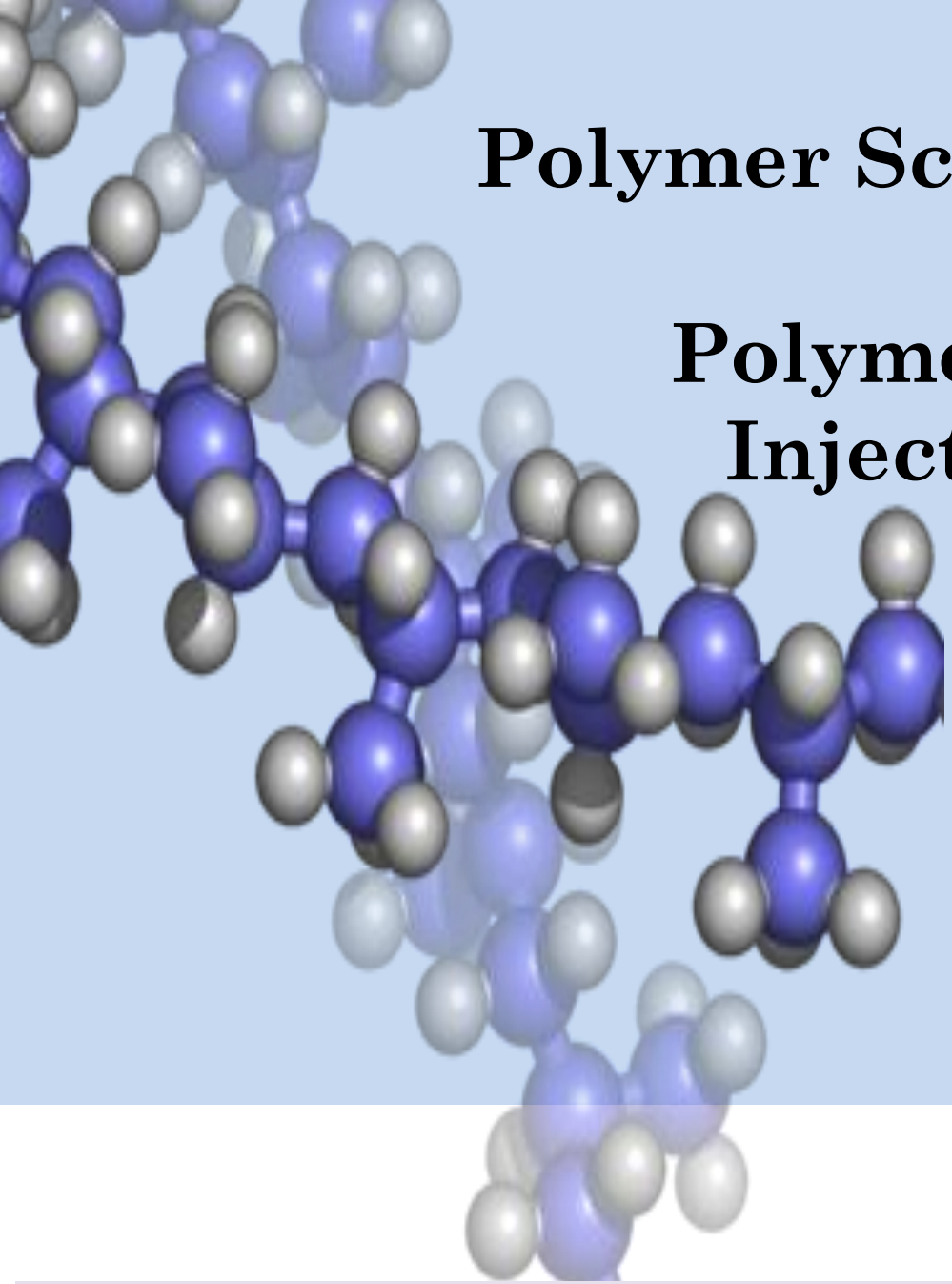


Polymer Science & Engineering

Polymer Processing: Injection Molding

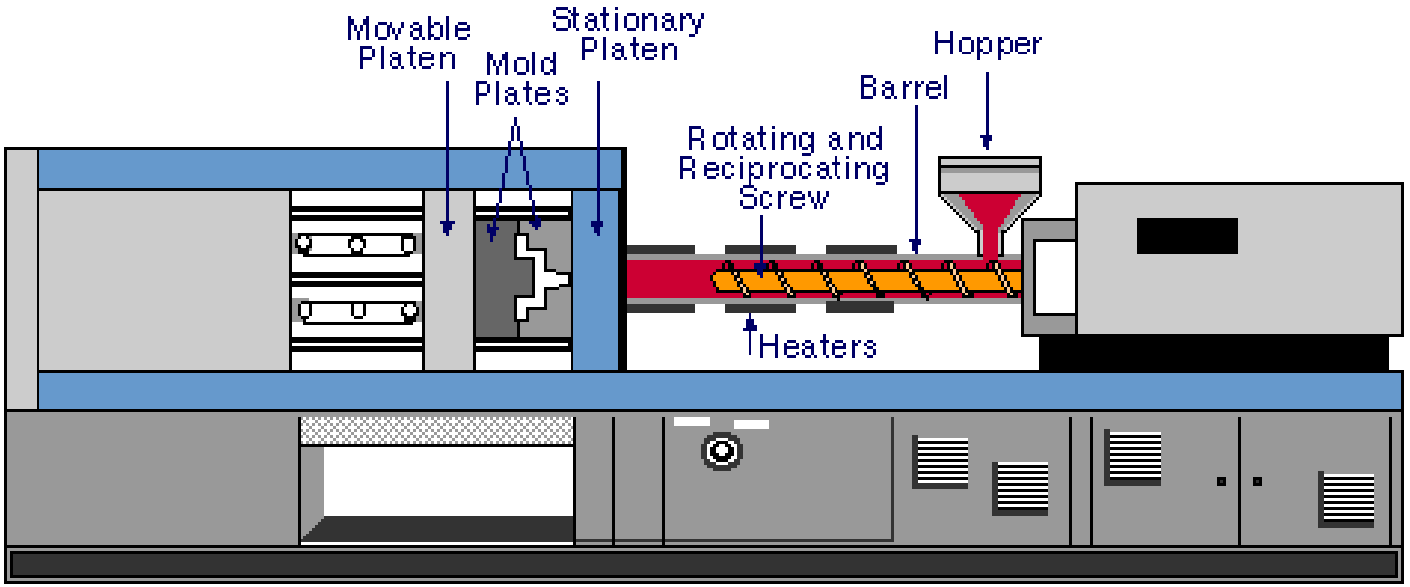
Dr. Motasem Saidan

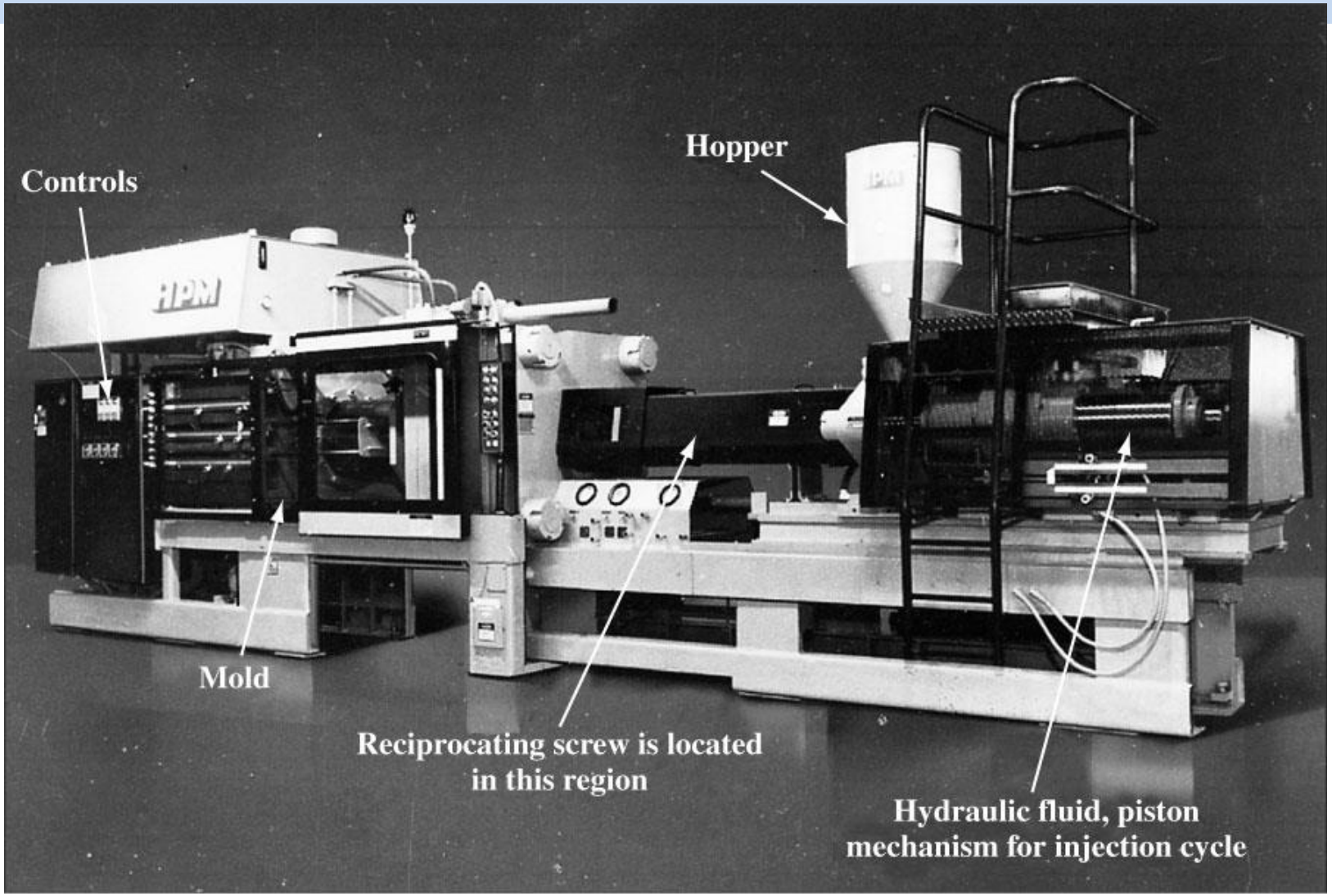
M. Saidan@gmail.com

Injection Molding

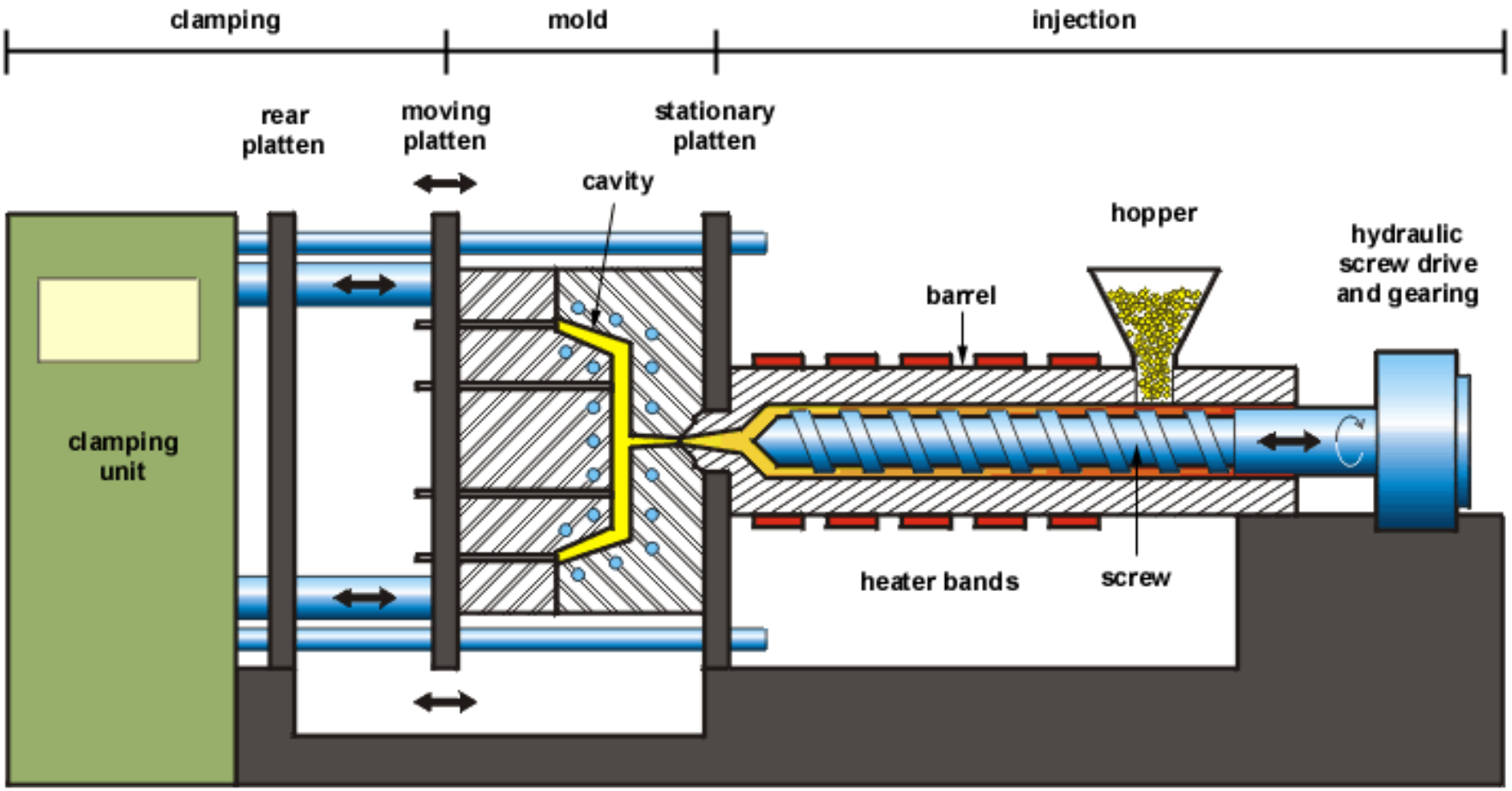
- Injection molding is a manufacturing technique for making parts from thermoplastic and thermoset materials
- In contrast to the extrusion (which makes continuous parts of constant cross section), injection molding make discrete parts (with complex and variable cross section)
- Injection molding is a cyclical steps process
- Molten plastic is injected at high pressure into a mold,
- The mold shape is the inverse of the desired shape.
- The mold is made from metal, usually either steel or aluminium
- The injection molding machine must, therefore, perform essentially three functions:
    1. Melt the plastic so that it can flow under pressure.
    2. Inject the molten material into the mold.
    3. Hold the melt in the cold mold while it solidifies and then eject the solid plastic.

Injection molding machine




Controls
Hopper
HPM
Mold
Reciprocating screw is located in this region
Hydraulic fluid, piston mechanism for injection cycle

Schematic of thermoplastic Injection molding machine



schematic of thermoplastic injection molding machine

Plastic Injection Molding

The key to success in injection molding are to have;

- Proper machine for good melting and injecting of the resin
- The proper resin to appropriate part performance
- A good mold for part definition and removal
- Proper operation for efficient molding cycle (mold cycle depends on the design of the mold and manufacturing parameters)

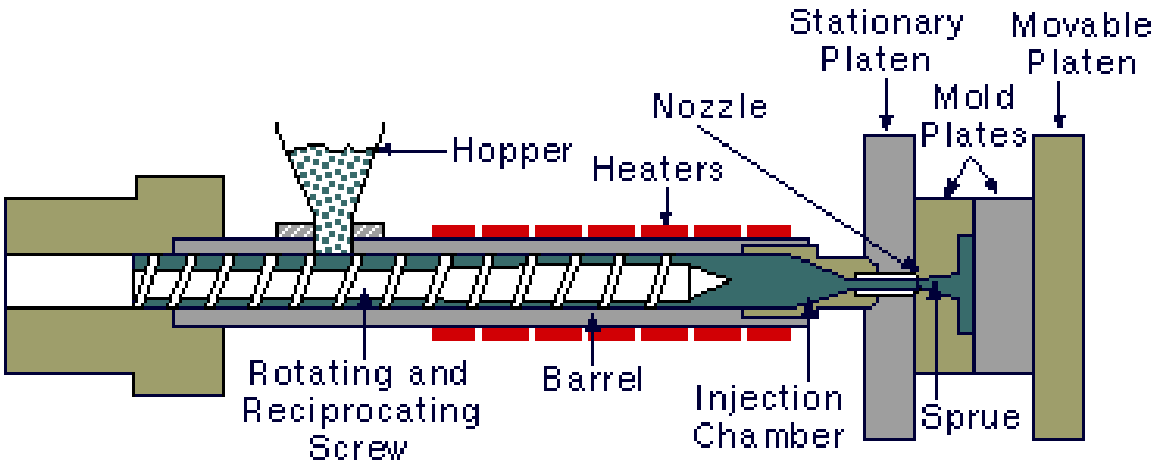
Process Operation

- Temperature: barrel zones, tool, mold zone
- Pressures: injection max, hold
- Times: injection, hold, tool opening
- Shot size: screw travel

Injection molding machine basics

- **The injection unit** essentially has two functions: melt the pellet or powder and then inject the melt into the mold. It consists of the hopper, a device for feeding process material; a heated cylinder or chamber where the material is melted; and a device for injecting the molten material into the mold.
- **The clamp unit or press end** of the injection molding machine performs three functions: opens and closes the mold at appropriate times during the molding cycle; ejects the molded part; and provides enough pressure to prevent the mold from opening due to the pressure developed in the mold cavity as it is filled with the melt by the injection unit.

The halves of the mold are attached to the platens, one of which is stationary and one of which moves as the clamp mechanism is opened or closed. Molds are generally designed so that the ejection side of the mold (mold core) is on the movable platen and the injection side of the mold (mold cavity) is on the stationary platen, which must provide an entry for the nozzle of the plasticizing chambers.

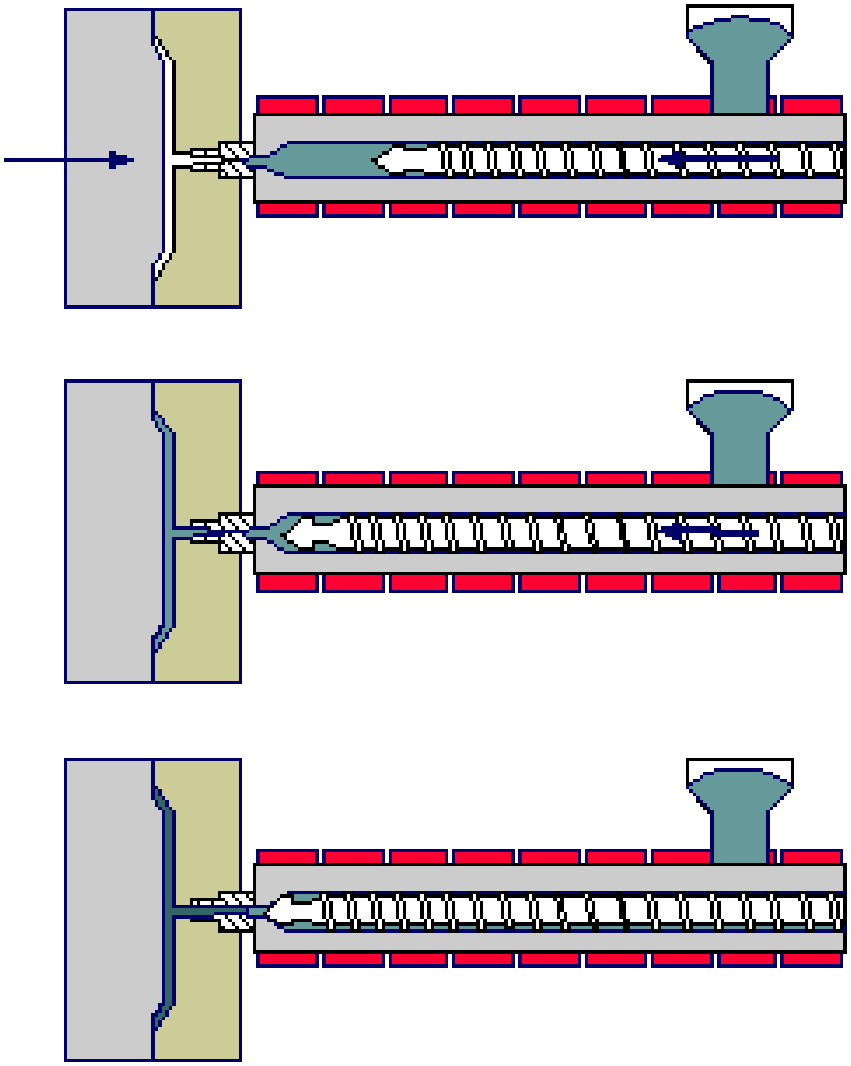


Injection Unit

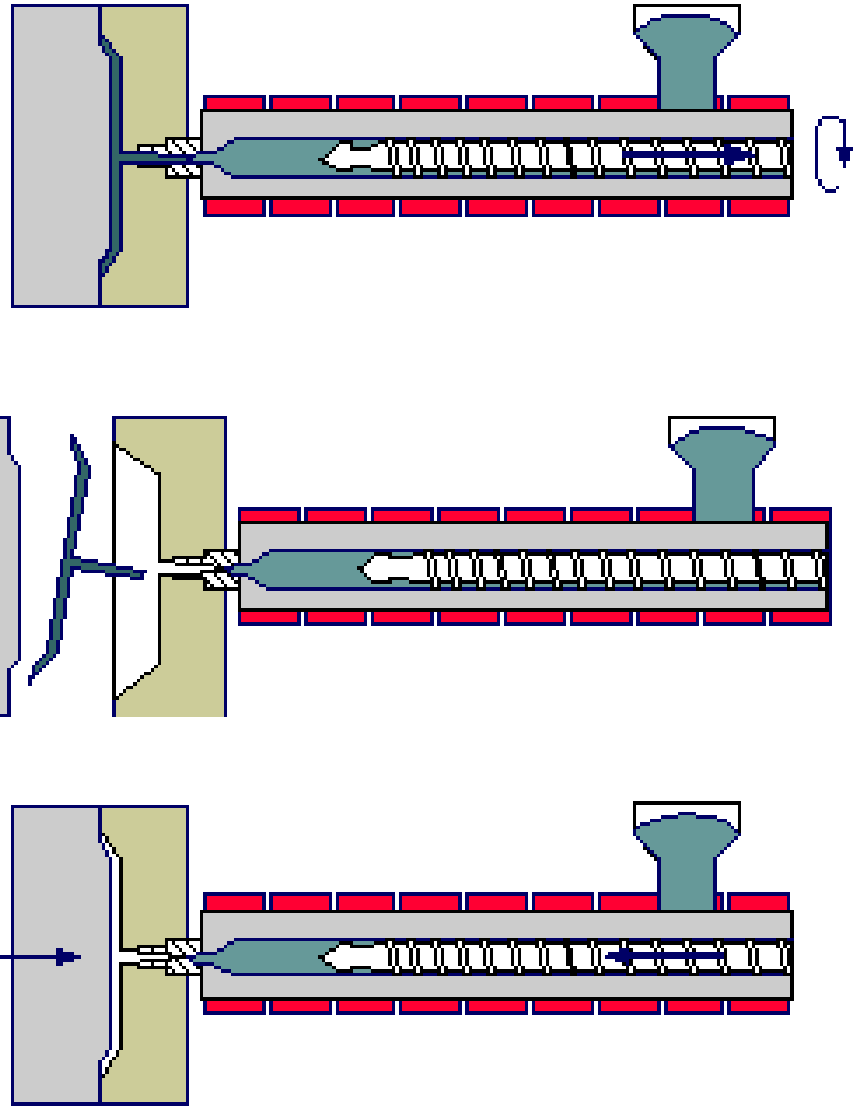
- Design of screw- similar to an extrusion screw
- 3 sections;
  - Feed section- to advance the resin
  - Compression section- to melt the resin
  - Metering section- to homogenize the resin and pump it forward
- The screw of injection molding machine is shorter than extruder, L/D ratios are 12:1 and 20:1
- Low L/D ratios suggest the mixing is less efficient in the injection molding machine
- The compression ratio (diameter of root at feed zone to the diameter of root at metering zone) often in the range of 2:1 and 5:1
- Low compression ratio means less mechanical action is added during melting process
- Important measure of the size of an injection molding is weight of resin that can be injected, called shot size. Typical shot size range from 20g to 20 kg
- Since shot size depends on the density of the plastic, PS has been chosen as the standard for rating the machine

Injection Molding Machine Operating Sequence

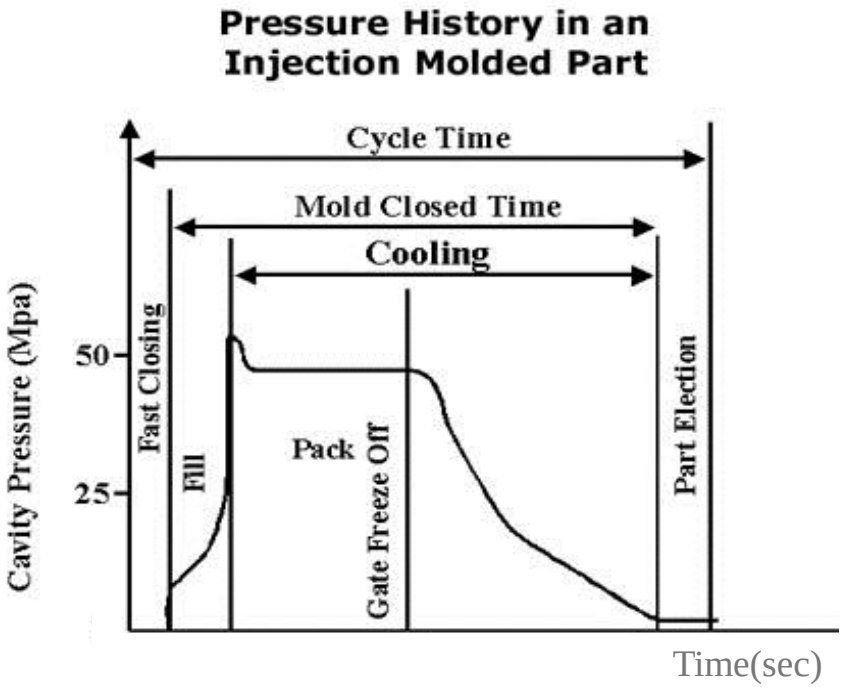
- Mold closes and screw begins moving forward for injection
- To ensure the resin does not flow backward, a check valve or nonreturn valve is attached to the end of screw
- Normally the screw will stay in the forward position, until resin began to harden in the mold
- The cavity fills as the screw moves forward
- The cavity is packed as the screw continues to move forward as part solidifies

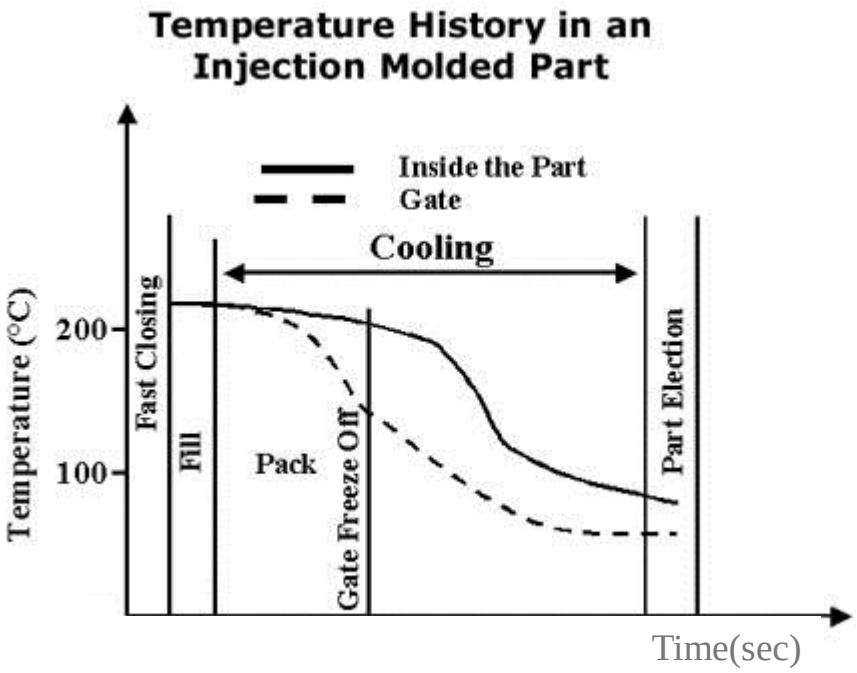


- The cavity cools as the gate freezes off and the screw begins to retract to plasticize material for the next shot
- Retraction of the screw, create space at the end of the screw
- Cooling of the part in the mold, until it can be removed
- While the part is cooling, the screw turns and melts additional resin
- The mold opens for part ejection
- The mold closes and the next cycle begins



Typical pressure/temperature cycle



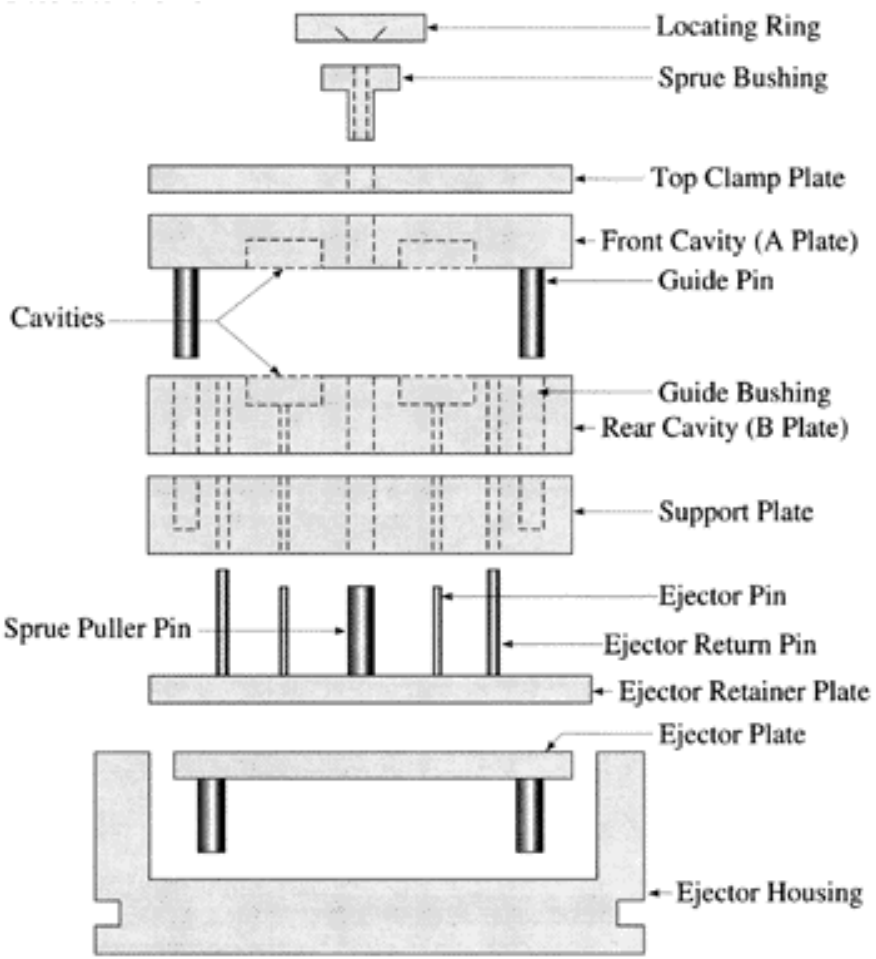


Cooling time generally dominates cycle time

$$t_{cool} = \frac{(\text{half thickness})^2}{\alpha}$$

$$\alpha = 10^{-3}\, cm^3/\text{sec} \quad \text{for polymers}$$

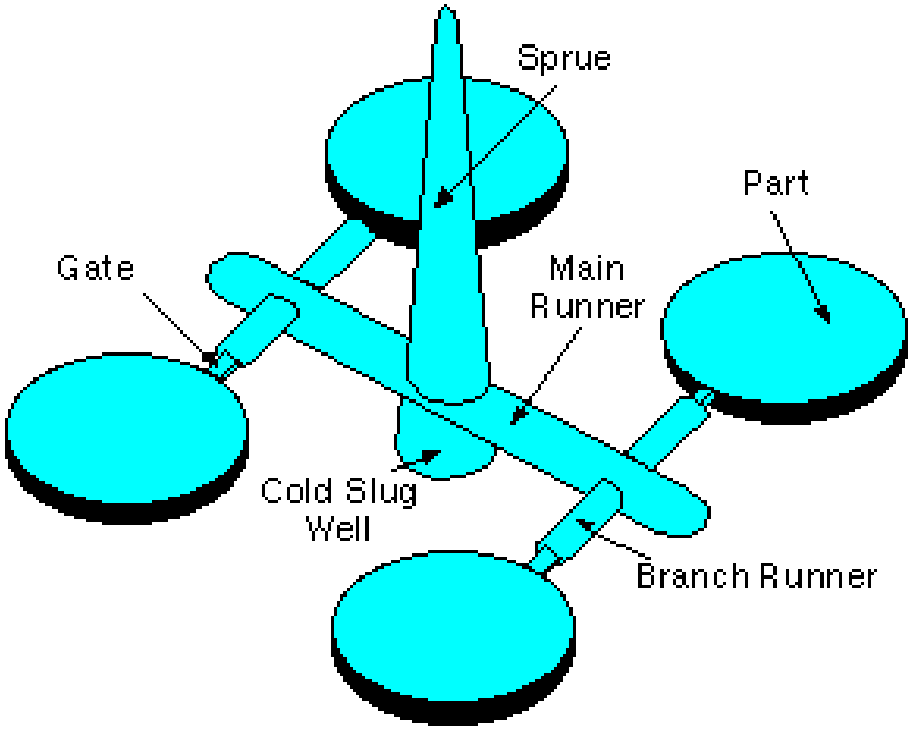
Molds


Locating Ring
Sprue Bushing
Top Clamp Plate
Front Cavity (A Plate)
Guide Pin
Cavities
Guide Bushing
Rear Cavity (B Plate)
Support Plate
Ejector Pin
Sprue Puller Pin
Ejector Return Pin
Ejector Retainer Plate
Ejector Plate
Ejector Housing

Mold Parts

- The connection from the injection unit to the mold is through the nozzle
- The channel that run through the stationary plate of the mold is called the sprue channel (material that is in the channel is called the **sprue**
- The solid sprue is removed from the finished part assembly after the part is ejected from the mold
- Resin flow from the sprue through **the runner** (connecting channel) to the mold cavities
- Flow characteristics (viscosity), temperature and other factors are important in determining the runner diameter and length.

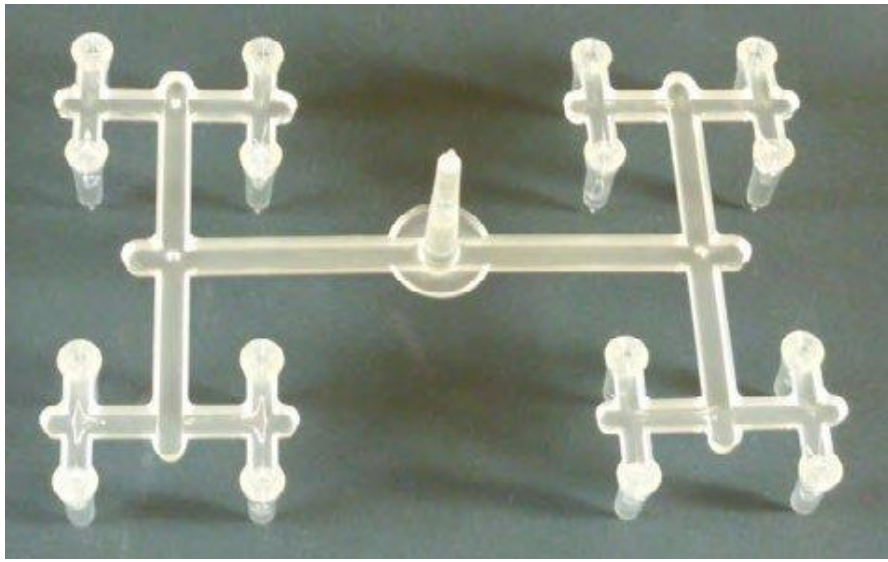
- If the diameter of the runner is too small or the length is too long, the resin can freeze in the runner before the mold is completely full
- If the runner system is too large, excess material would be ejected
- If the resins have a high viscosity, larger runners are needed compared to low viscosity resin
- The optimum flow of the resin through the runner system depends on the shape and diameter of the channel

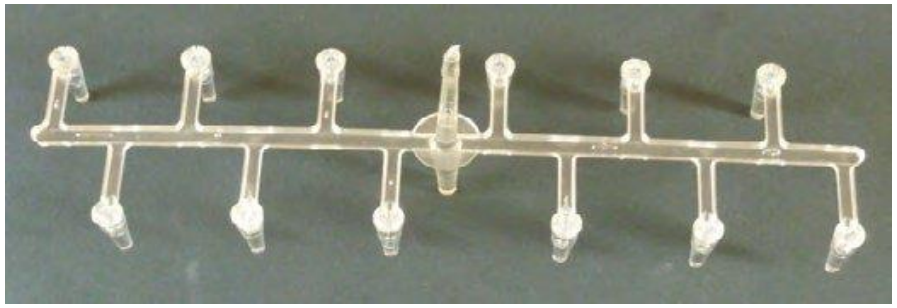


Mold parts

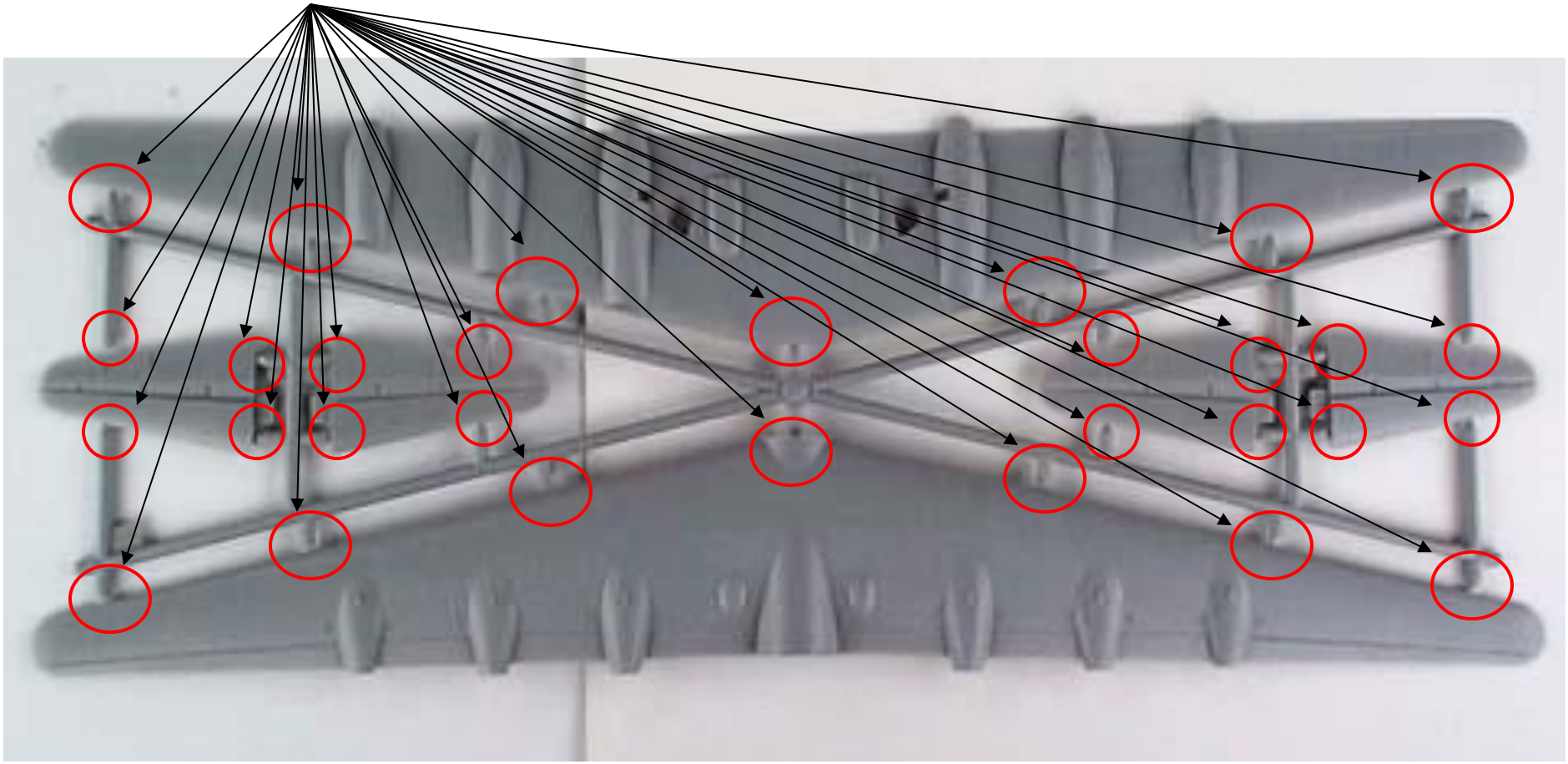
- Secondary runner channel are used for multicavity molds
- The flow into the secondary channel should be streamlined (angle in flow direction)
- The streamlined minimizes shear on the resin

- **The Gate** is the end of runner and the entry path into the cavity
- The gate shape can also affect the filling of the cavity, dimension and properties of the parts
- Gate is the most restricted point in injection molding system, i.e. for reinforcement and filler + polymer systems
- It narrows to a point as it moves from the runner to the cavity
- The advantage; separation of the parts and the runner is automatic
- Disadvantage; gate cannot be used for some resins because of high shear



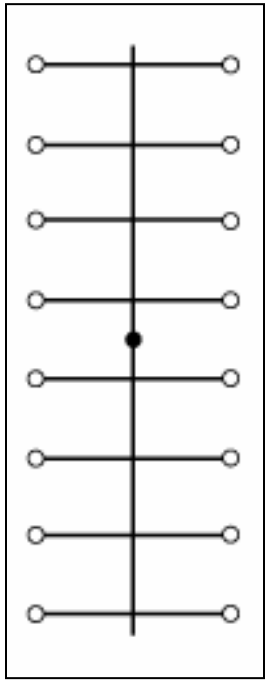


Gates

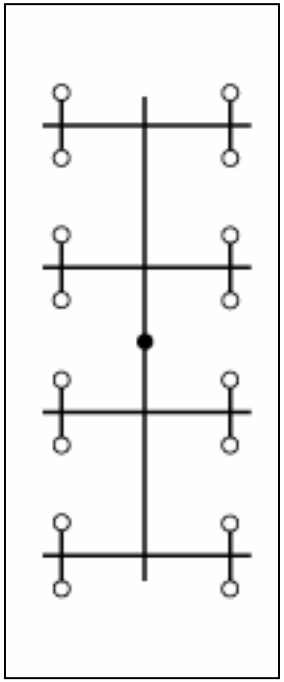


Runner Layout

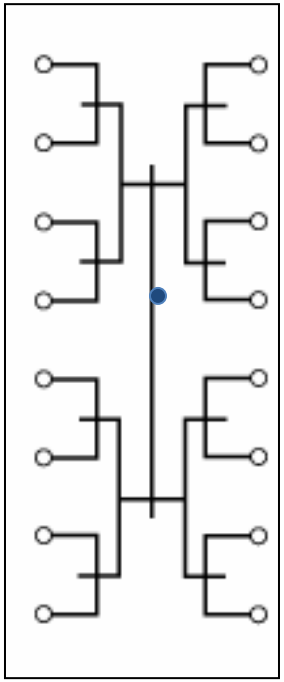
Conventional



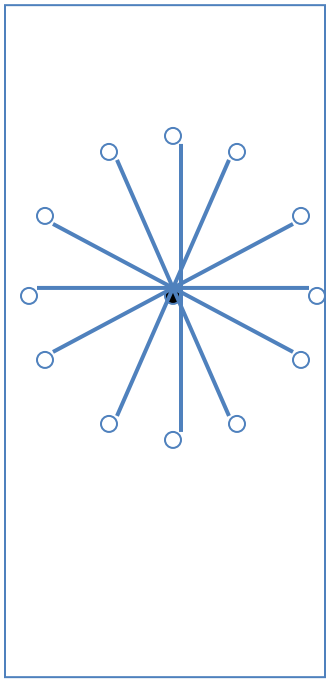
Improved



Balanced “H”

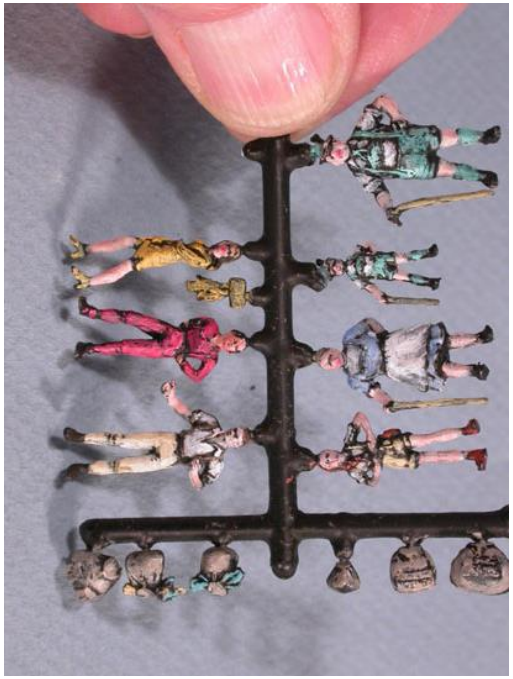


Circular



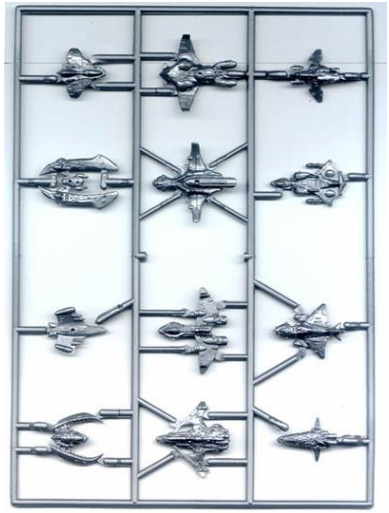
Runner Layout

Conventional



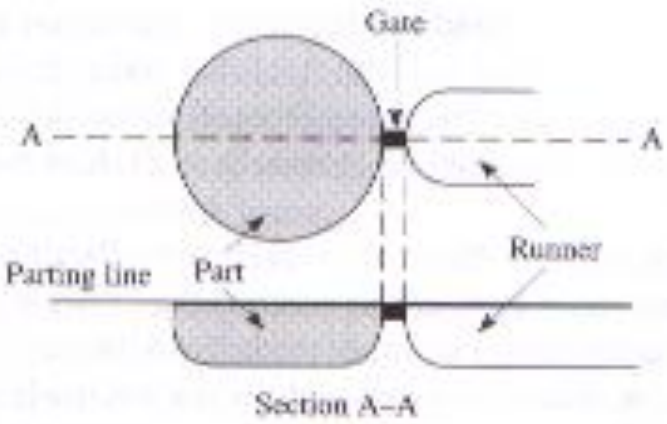
Improved

Balanced “H”

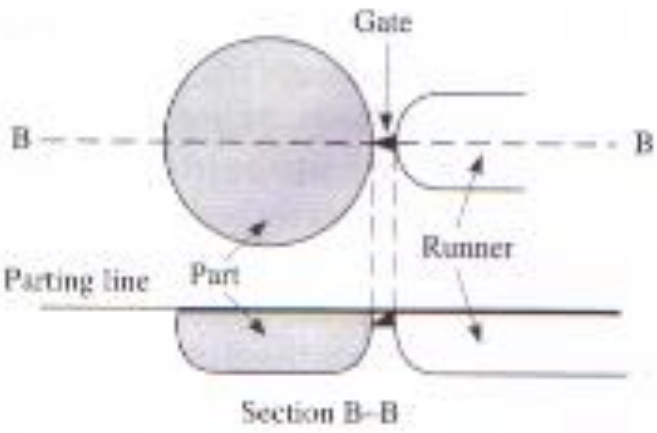


Circular

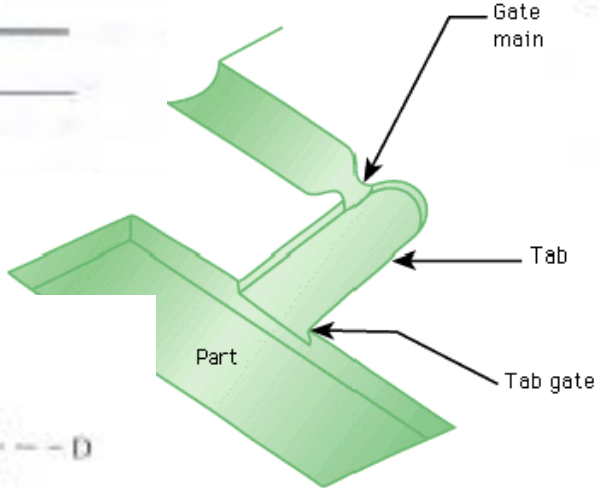
Gate design and types

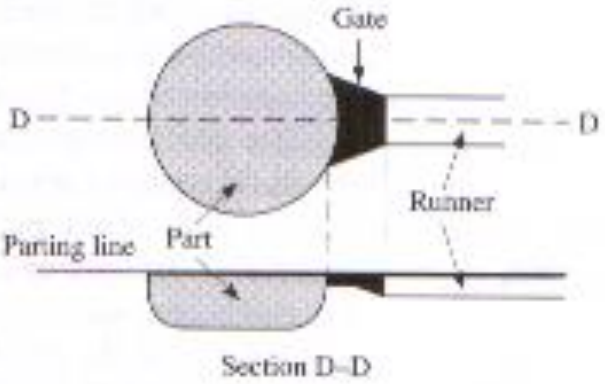


Edge gate

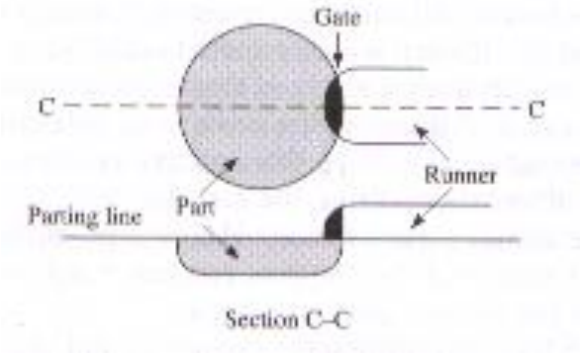


Submarine Gate





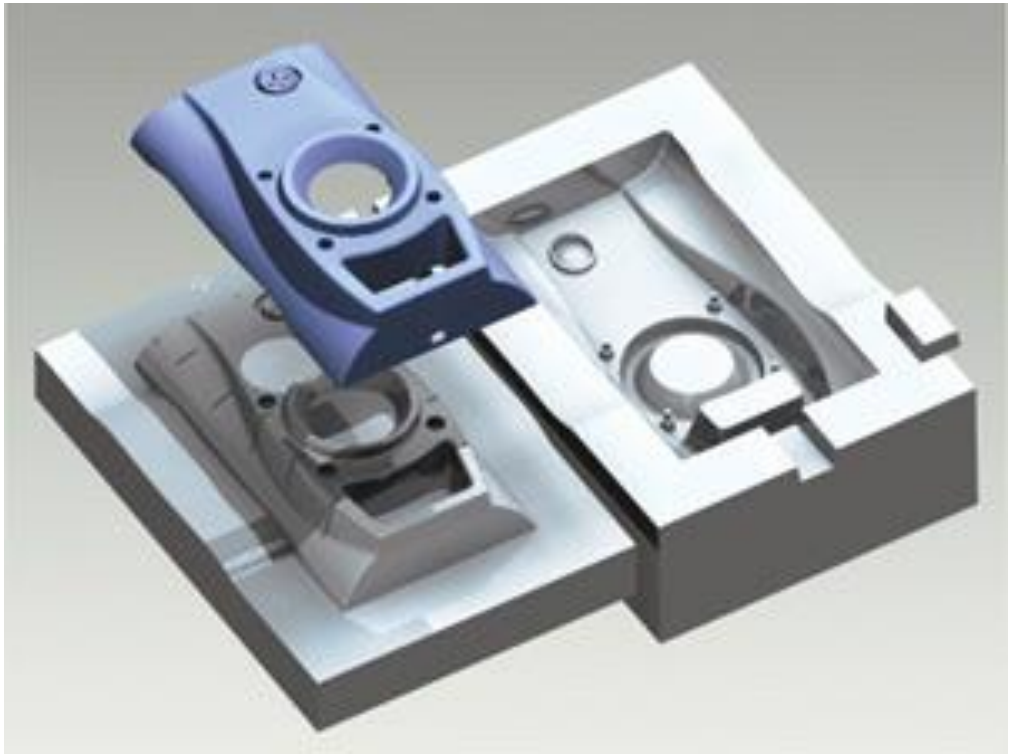
Fan Gate



Tap Gate

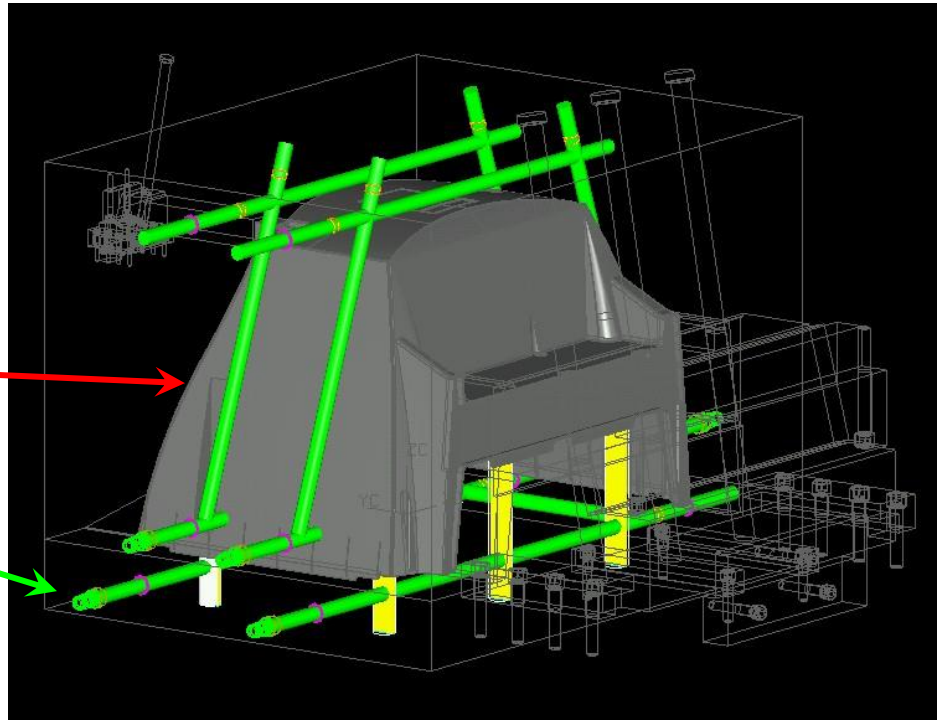
Cavities

- Are actual molding locations
- Resin enter the cavities through gate, fills the cavities, and cools to form the solid. The parts are ejected and finished
- Cavities are the heart of the molding process, and must be precisely prepared
- The shape of the cavities determines the shape of the part

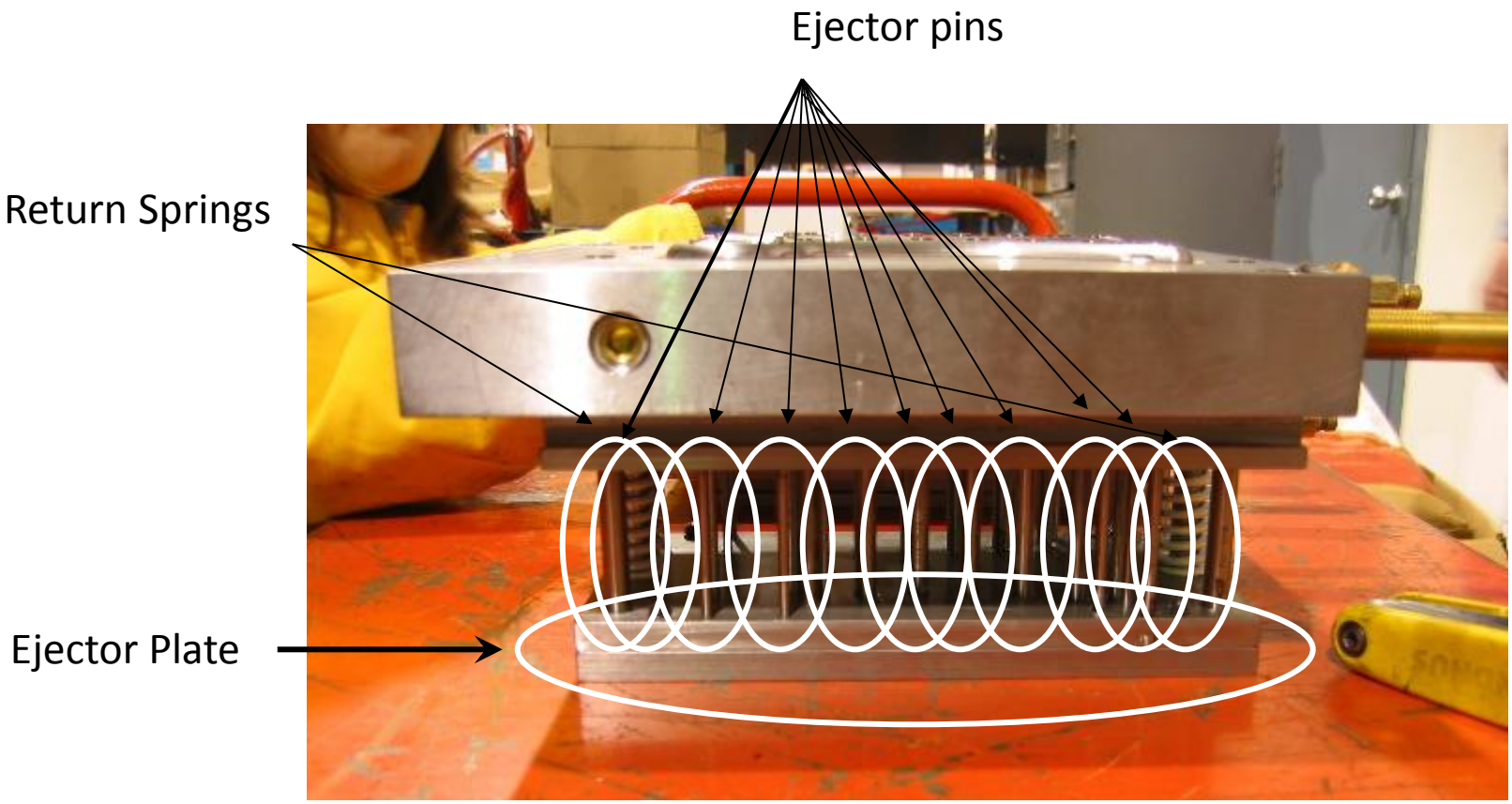


Cooling

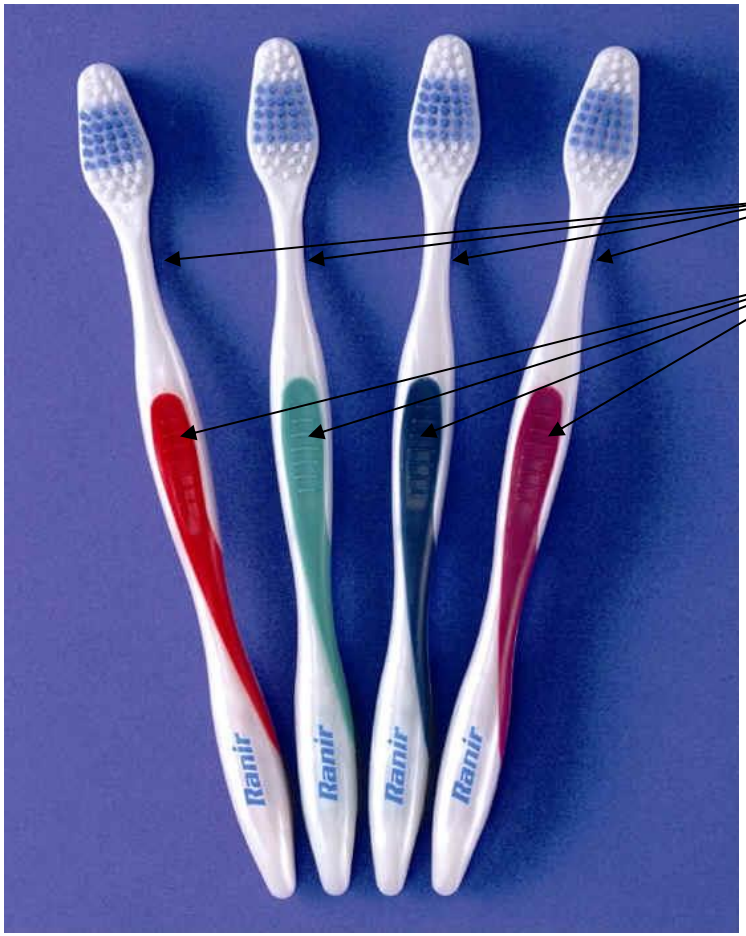
- Reduce cycle time
- Control formation of crystalline areas
- Freeze part



Ejector system

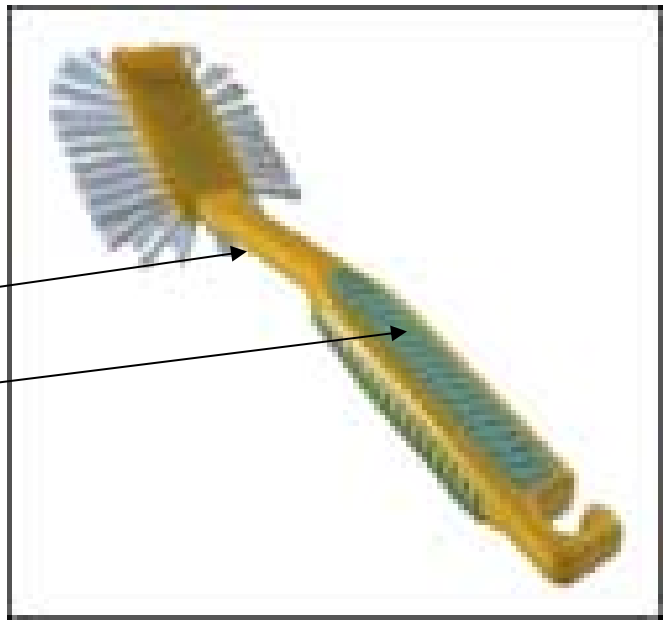


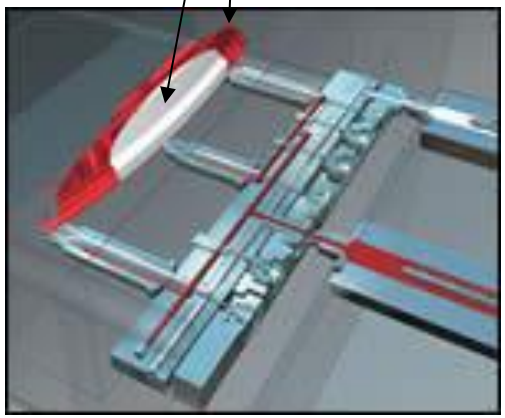
Co-injection molding



Part 1

Part 2

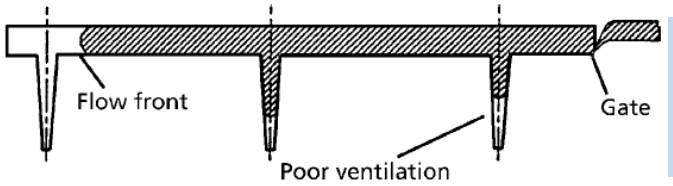




Injection Molding Defects

1. Short shots
2. Mold flash
3. Sink marks
4. Voids
5. Flow marks and weld lines
6. Brittleness
7. Grooves & Jetting
8. Air streaks
9. Moisture streaks
10. Color streaks
11. Stress whitening
12. Burnt streaks
13. Excessive shrinkage
14. Gloss difference
15. Sticking in mold

1. Short shot



Short shots appear when an injection mold has not been properly filled. As a result, parts appear incomplete. This effect usually appears far from the gate, especially where long flow distances are involved, thin walls (or a combination), or thin ribs to be filled by the polymer.

The effect of short shots may also appear in other areas of the molding and often the reason can be traced to gate design.

Common causes for short shots can be:

- Insufficient amount of material injected
- Insufficient injection fill speed
- Insufficient pack pressure
- Insufficient injection time
- Unbalanced multiple cavity mold
- Foreign material clogging nozzle
- Stock temperature too low
- Runners, gates, sprue or vents too small
- Mold temperature too low
- Excessive wear of screw or barrel

2. Mold flash

Parts exhibiting mold flashes are characterized by film-like plastic edges attached to the molded parts. The flashes often appear near commissures, sealing faces, venting channels or ejectors

Common reasons for mold flashes can be:

- Excessively high injection pressure (fill or pack)
- Low clamping force pressure
- Stock temperature too high
- Resin melt viscosity too low
- Shut-off faces of mold mismatched

3. Sink marks

Sink marks are aesthetic defects at the surface of the molding. They appear as depressions and typically appear at thicker sections of the molding.

The root cause is usually shrinkage of the PP material during the crystallization process.

Possible causes for this excessive shrinkage can be:

- Insufficient injection pressure
- Insufficient dwell or hold time
- Insufficient amount of material
- Insufficient cooling or injection time
- Excessively high injection speed
- Melt or mold temperature too high
- Poor part design, non-uniformed walls and/or excessive wall thickness
- Improper gate location or design

4. Voids



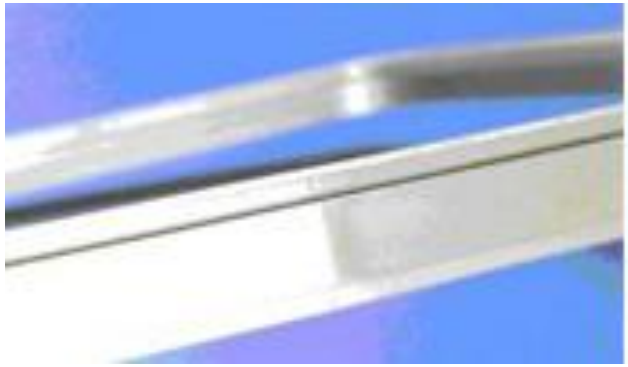
Voids appear as small bubbles or blisters within the molding and are typically created by entrapped air, gas, or perhaps a vacuum.

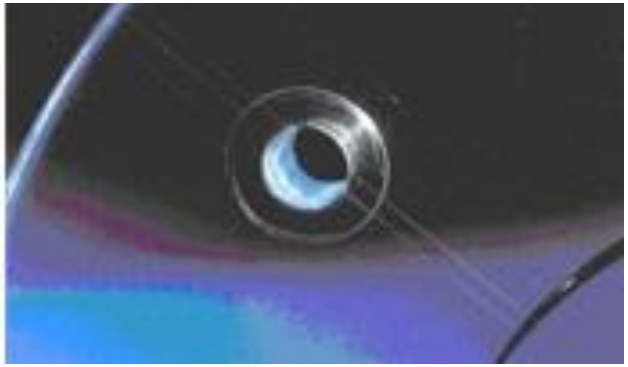
Voids become more apparent if the item is highly transparent.

Possible causes can be:

- Failure to fill mold completely
- Poor venting of mold, particularly around projections
- Improper gate location
- Fill rate too rapid (trapped air produces short shots)
- Mold temperature mismatched
- Excessive part thickness (greater than 6.3 mm (1/4”)
- Entrapped moisture
- Entrapped air through porous or very fine additive powders absorbing air

5. Flow marks and weld lines





Weld lines represent an optical as well as mechanical defect in a molded part.

Weld lines typically appear in the area where the polymer flows come together during the injection process.

Flow marks appear due to flow turbulences during injection. Those effects are particularly visible in colored or metallic pigmented parts in the form of color changes.

Possible causes can be:

- Stock temperature too low
- Mold temperature non-uniform or too low
- Mold fill too fast or too slow
- Excess mold lubricant
- Scratched or dirty mold surface
- Fill speed and/or packing time too low
- Inadequate venting
- Improper gate location or design

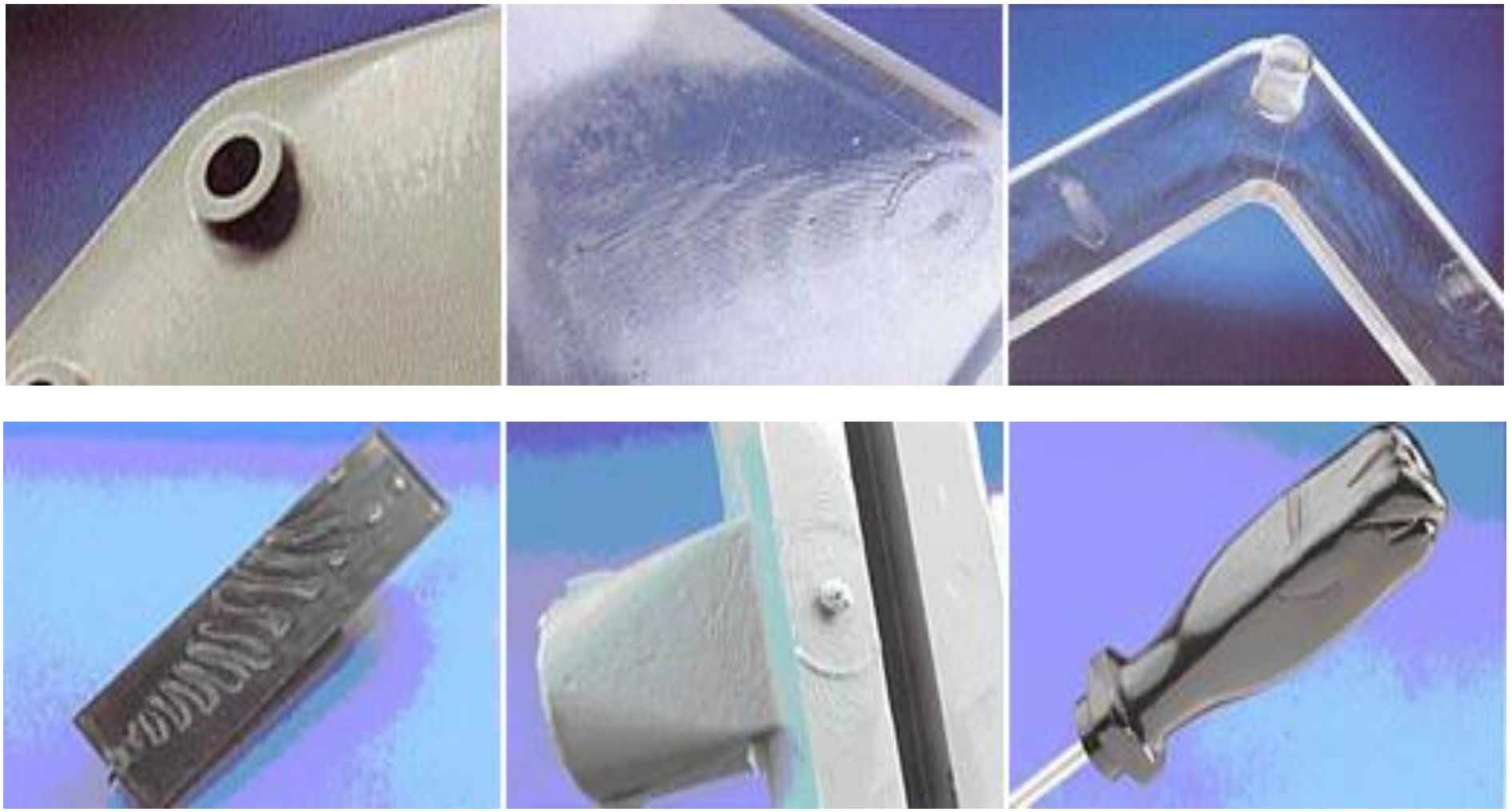
6. Brittleness

Sometimes following the injection process, parts may be produced which show crack formations when low stress or flexing is applied.

This phenomenon can be caused by:

- Degraded material from barrel (excessive melt temperature)
- Contamination by another material(s)
- Improper design (Inadequate radii at corners, notch or thread)
- Use of improper color concentrates (non-compatible carrier resin)
- Voids
- Stock and mold temperature too low
- Excessive amount of regrind
- Molded stress too high

7. Grooves & Jetting

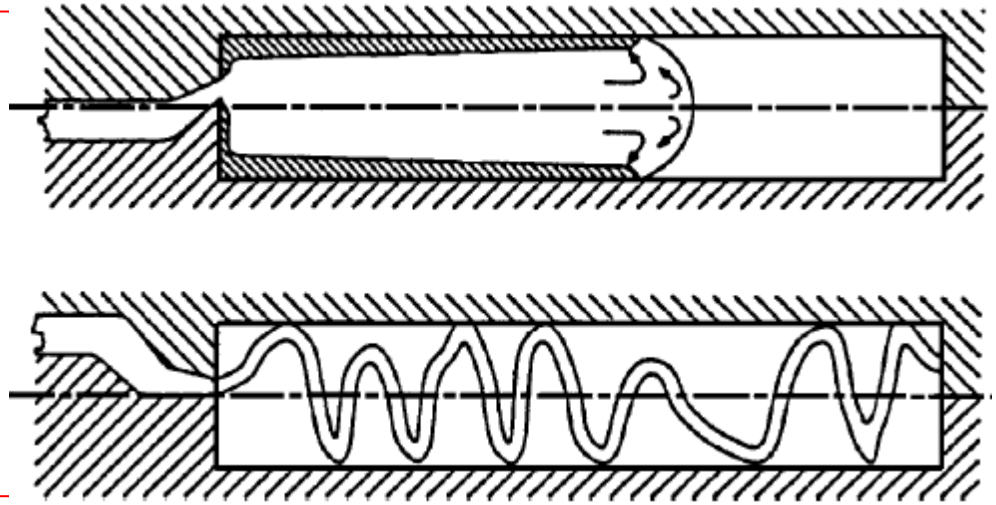


- Grooves are a surface defect where “rings” appear at the surface of molded parts mainly around pin point gates and concentrically spreading over the molding

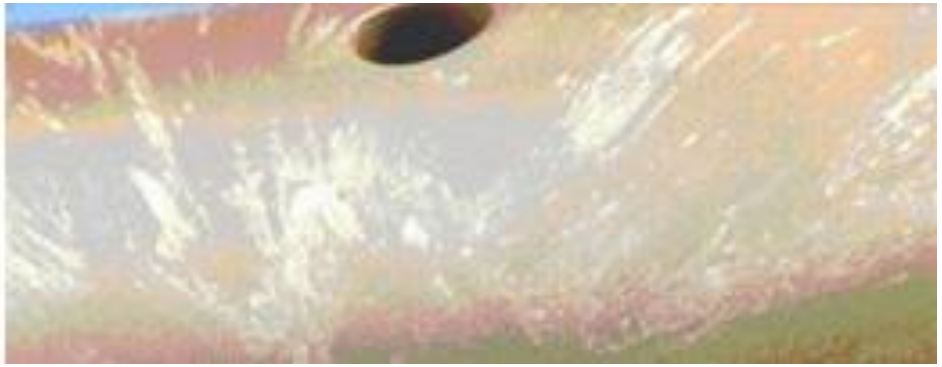
- Jetting is a similar defect to grooves. Also in this case rough or matt lines appear at the surface of the molding starting at the gate and spreading over the entire part. The lines will appear different in gloss and color and are more visible with colored parts.

Reason for this type of defect can be:

- Insufficient stock temperature
- Insufficient injection speed
- Mold temperature too low
- Improper gate location or design
- Undeveloped front flow
- Excessive injection speed



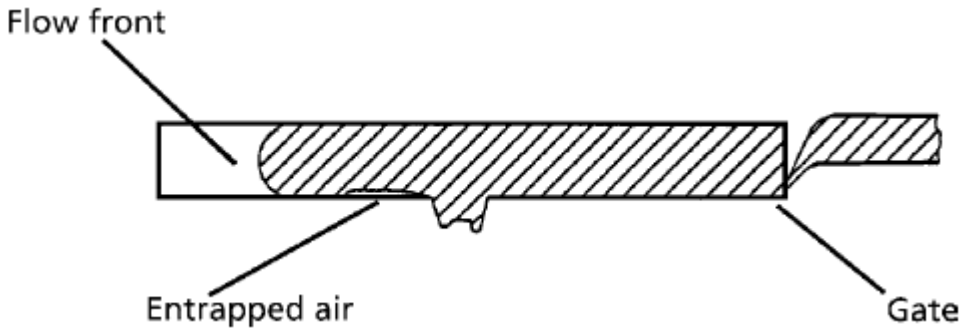
8. Air streaks



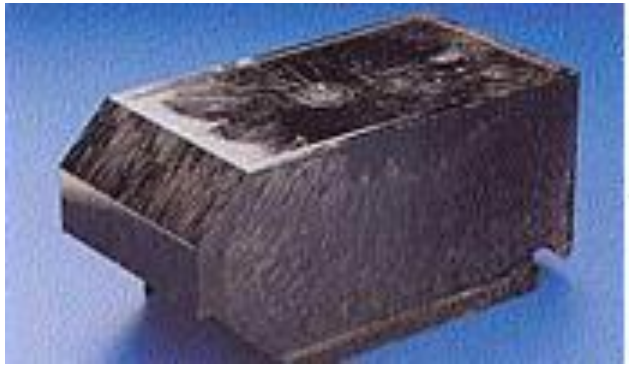
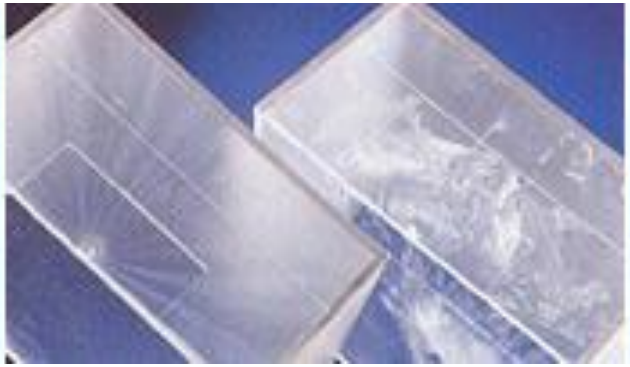
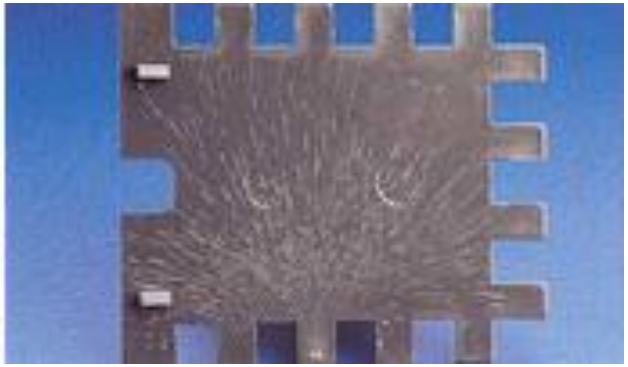
Air streaks in molded parts appear as matt, silvery or white lines (streaks) at the surface of the molded parts. They can usually be found near domes, ribs and where the wall thickness of the molded part may vary. They also can appear near the sprue or near engraving and depressions.

Reasons for this effect can be:

- Inadequate venting
- Excessive injection speed
- Air creates suck back
- Excessive moisture in material
- Mold temperature too low



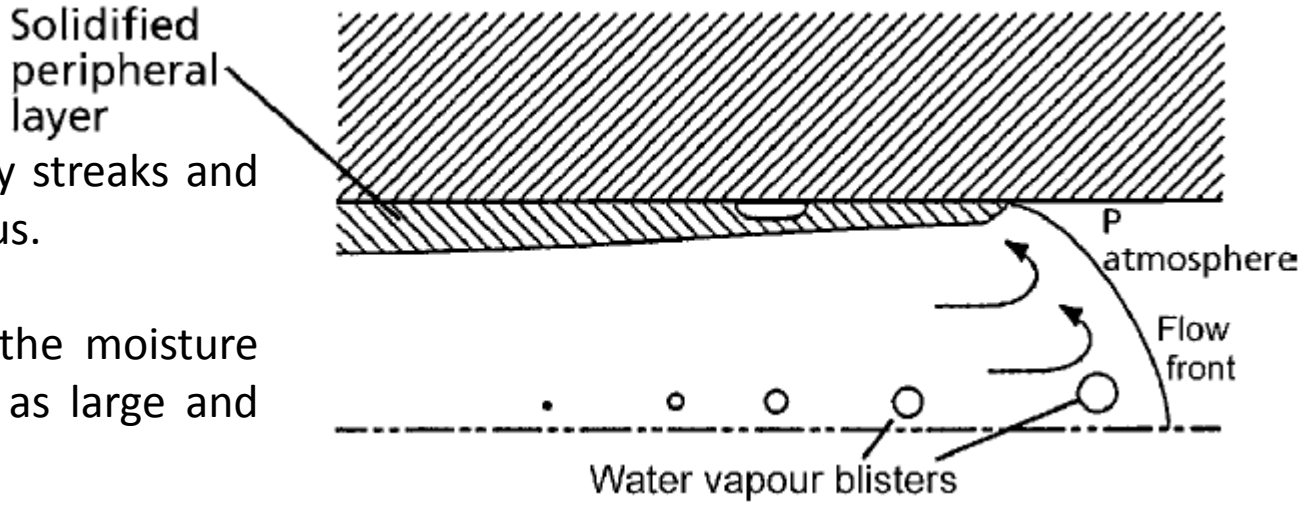
9. Moisture streaks

Moisture streaks can appear on the surface of molded parts as a U-shaped profile open against the direction of the flow.

Moisture streaks are typically caused by:

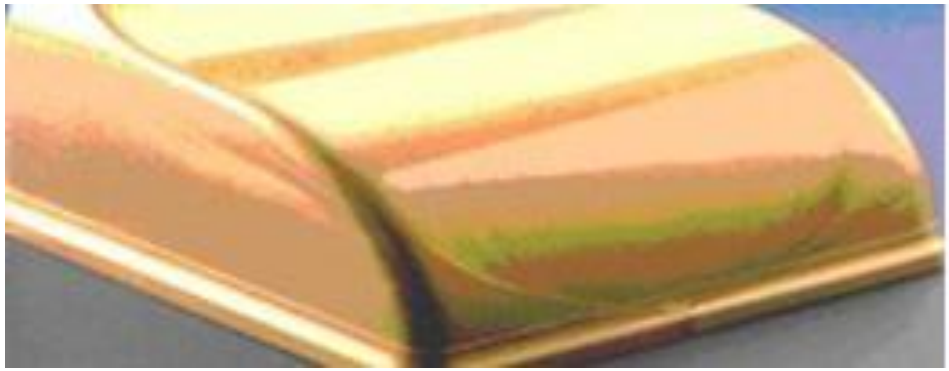
- Moisture in resin
- Moisture condensation on surface of mold



They usually appear as silvery streaks and their surface is rough or porous.

Moisture streaks caused by the moisture on the mold surface appear as large and dull lamellar structures.

10. Color streaks

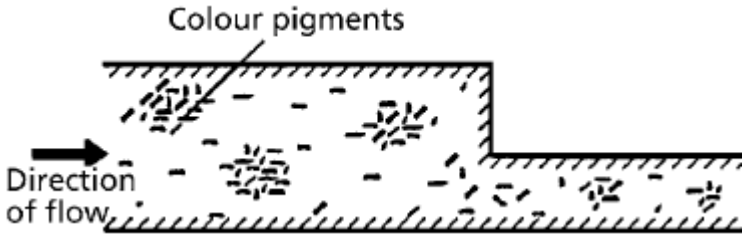


Color streaks can appear on colored parts as a result of uneven distribution of the color pigments in the item or different orientation of isotropic pigments in the molded item.

Thermal effects (degradation of pigments) can also cause different color shading visible on the item.

Color streaks can be created through:

- Inadequate color dispersion or distribution
- Improper masterbatch
- Improper design
- Lack of lubricant for dispersion
- Stock temperature too high
- Excessive injection pressure



11. Stress whitening

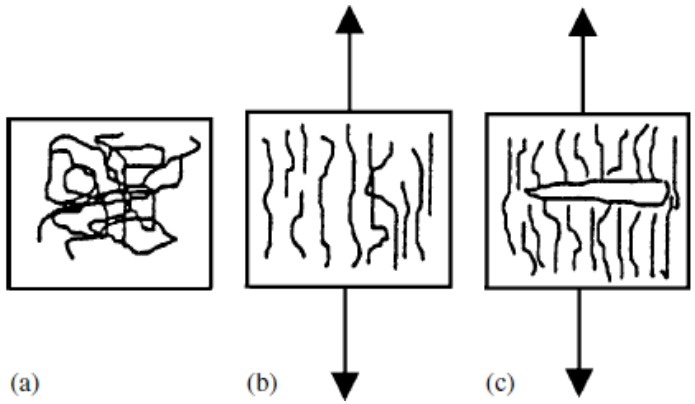


Stress whitening is caused by stress cracks in the polymer material due to internal as well as external stress.

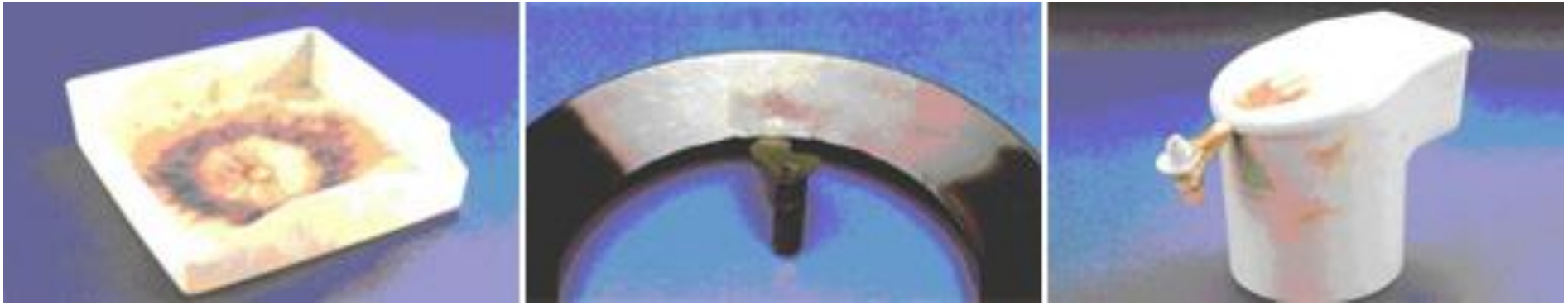
Typical appearance locations are de-molding points. Stress cracks caused by internal stress can often appear days or even weeks after production during the relaxation process of the molded part.

Stress cracks can be due to:

- Characteristic of individual plastic (Copolymer, impact resistance, etc.)
- Residual stress due to molding condition (e.g.: melt temperature, part design, resin flow behavior etc.)
- Chemical resistance (ESCR)



12. Burnt streaks



Burnt streaks are defects caused by degradation of the polymer during the injection process and appear as brownish or silvery discolorations at the surface of the molded article.

Burnt streaks can be caused by:

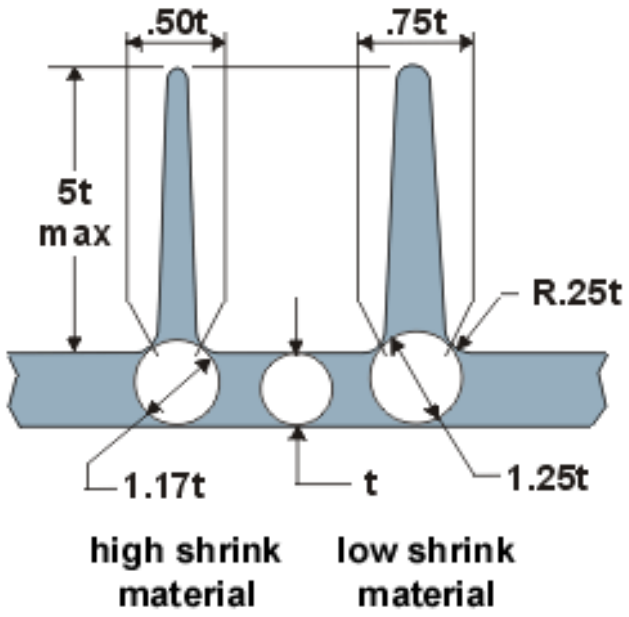
- Injection pressure too high
- Injection speed too high
- Back pressure too high
- Stock temperature too high
- Inadequate venting
- Improper gate location or design
- Other material contamination

13. Excessive shrinkage

Excessive shrinkage is a characterized by molded parts which appear smaller than the intended dimensions

Excessive shrinkage can be caused by:

- Cure time too short
- Pack pressure too low
- Mold or stock temperature too high
- Insufficient injection pressure
- Runners or gates too small
- Poor part design, varying wall thickness



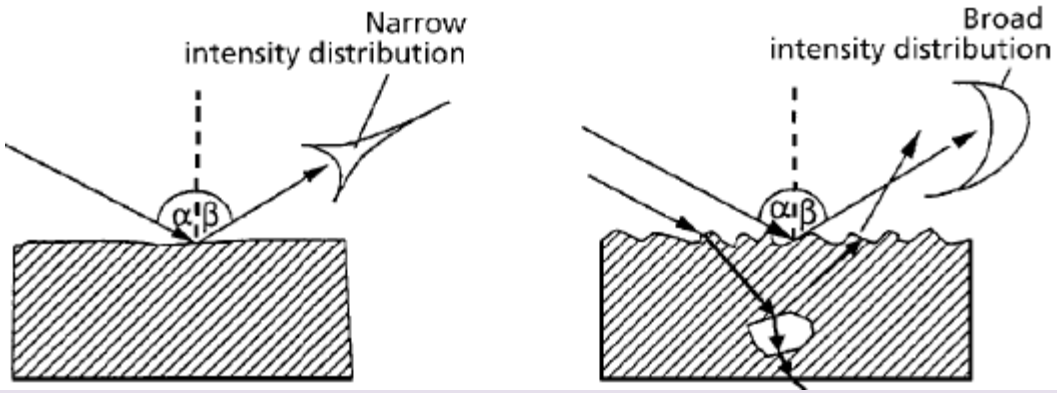
14. Gloss difference



Gloss differences typically appear due to wall thickness differences of the molded parts and subsequently different rates of cooling at different areas.

Gloss differences can be caused by:

- Flowability at surface of mold
- Processing temperature
- Variable cooling conditions
- Stress residue in mold



15. Sticking in mold

Molded parts may stick in the mold and cannot be ejected properly.

This typically occurs if the shrinkage level is too low or too high to allow easy ejection, or the shape and surface finish of molded articles creates a “sticking” effect in the mold itself.

Sticking in the mold can be due to:

- Overpacking – injection pressure too high
- Underpacking – excessive shrinkage
- Insufficient cooling
- Highly polished core surface > draw polish
- Insufficient knockout action
- Surface irregularities in the mold

Valuable Book

A copy will be provided for self reading