POLYMER PROCESSING

MODULE - III

INJECTION MOLDING

Module 3:

Injection moulding –Moulding cycle.

Machine construction –barrel, screw, nozzles, clamping system,

Machine ratings,

Basic mould construction –classification, sprue, runner, gate systems, mould cooling, ejection,

Part cooling analysis,

Effect of process variables on product quality.

Special Injection Mouldings.

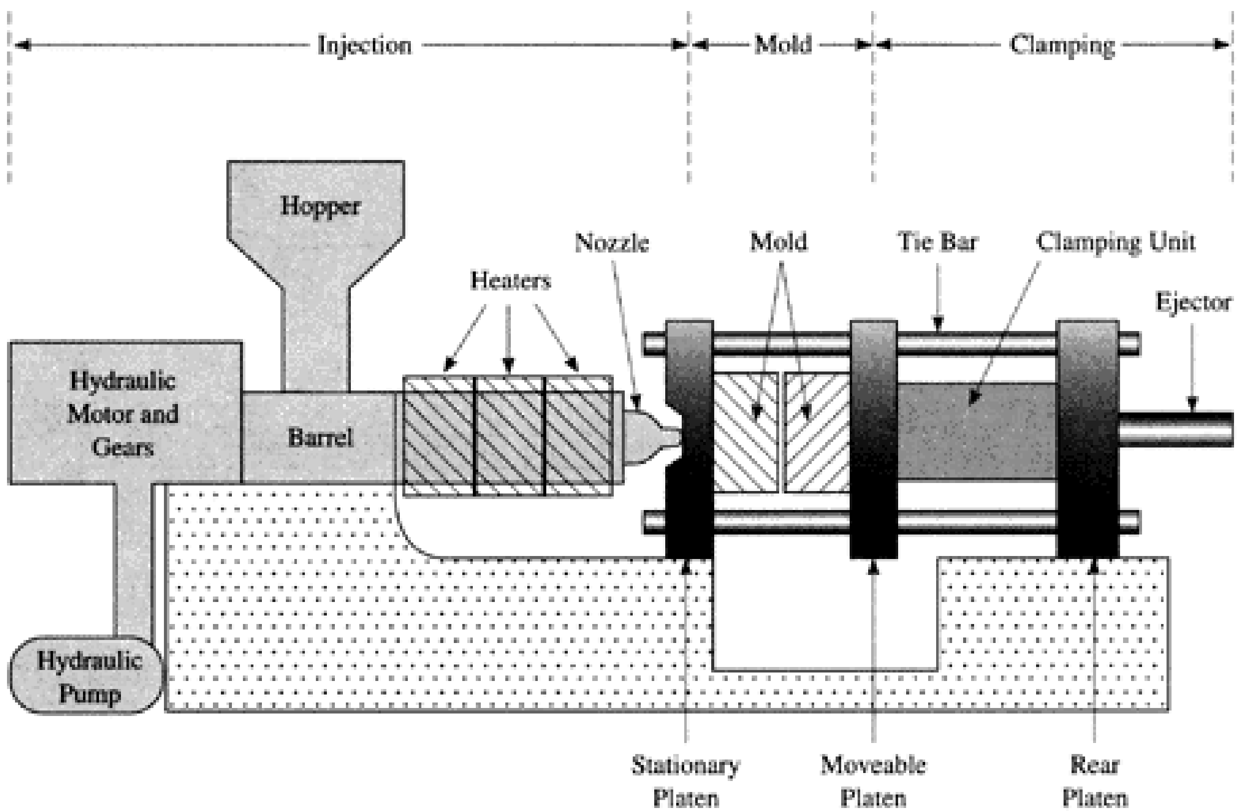
Product defects and its remedies. [7]

INJECTION MOLDING

•It is the predominant process for fabrication of thermoplastics into finished forms, and is increasingly being used for thermosetting plastics and fiber filled composites.

•Products ranging in weight from 5g – 85 kg and it is estimated that 25% of all thermoplastics are injection molded.

•The objective of injection molding is to obtain minimum cycle time with minimum scrap and to minimize production cost.



INJECTION MOLDING

The advantages of injection molding are:

- Parts can be produced at high production rate
- Large production volume is possible. • Relative low labor cost per unit.
- Process is highly susceptible to automation
- Parts require little or no finishing
- Most economical way to fabricate for complex shapes
- Close dimensional tolerances can be maintained.

The disadvantages of injection molding are:

- Intense industry competition often results in low profit margins
- Mold cost are high and hard to design
- Molding machine and auxiliary costs are high
- Lack of knowledge about the fundamentals of the process and properties of materials may cause problems.

In principle, injection moulding is a simple process. A thermoplastic, in the form of granules or powder, passes from a feed hopper into the barrel where it is heated so that it becomes soft. It is then forced through a nozzle into a relatively cold mould which is clamped tightly closed. When the plastic has had sufficient time to become solid the mould opens, the article is ejected and the cycle is repeated.

INJECTION MOLDING

- **Classification of injection molding are:**
- **<u>Single stage plunger machine:</u>**

A predetermined quantity of moulding material drops from the feed hopper into the barrel. The plunger then conveys the material along the barrel where it is heated by conduction from the external heaters.

The material is thus plasticised under pressure so that it may be forced through the nozzle into the mould cavity. In order to split up the mass of material in the barrel and improve the heat transfer, a torpedo is fitted in the barrel.

The disadvantages of this type of machines are:

(a) There is little mixing or homogenisation of the molten plastic.

(b) It is difficult to meter accurately the shot size.

(c) The presence of the torpedo causes a significant pressure loss.

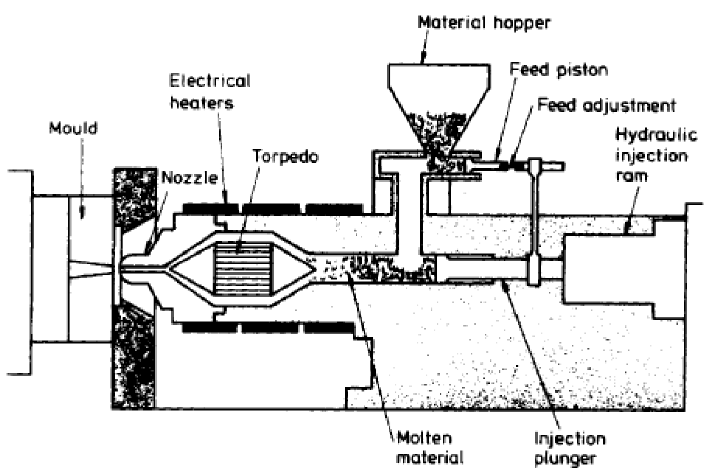


Fig. 4.30 Plunger type injection moulding machine

INJECTION MOLDING

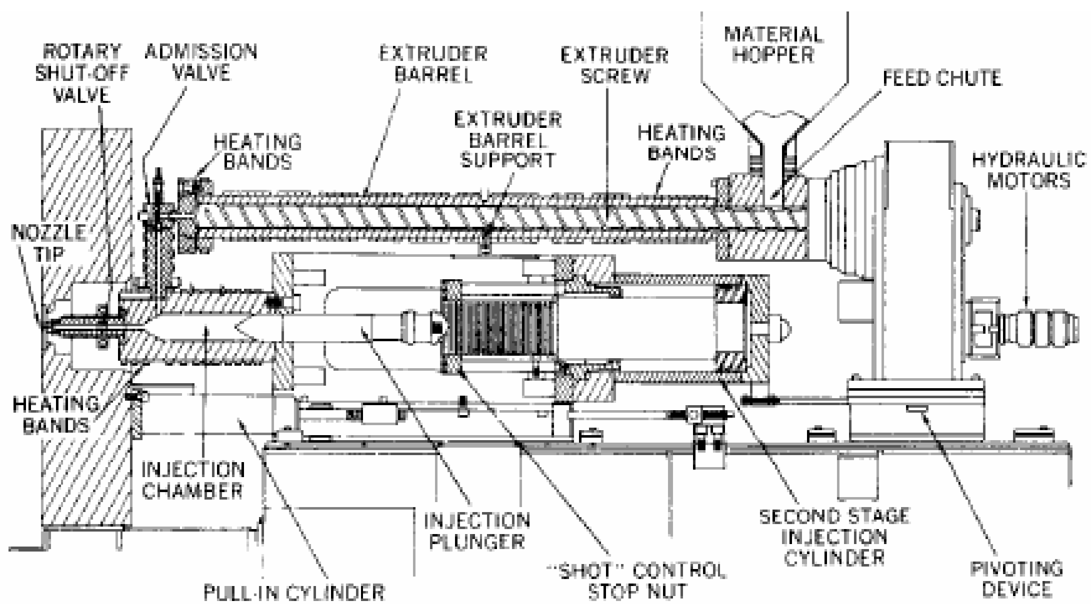
- **Classification of injection molding are:**
- **Two stage Screw - plunger machine:**

This type of machine has two barrels.

Raw material is fed into the first barrel where an extruder screw or plunger plasticises the material and feeds it through a non-return valve into the other barrel.

A plunger in the second barrel then forces the melt through a nozzle and into the mould. Another advantage is that there is no longer a need for the torpedo on the main injection cylinder.

However, nowadays this type of machine is seldom used because it is considerably more complicated and more expensive than necessary.



INJECTION MOLDING

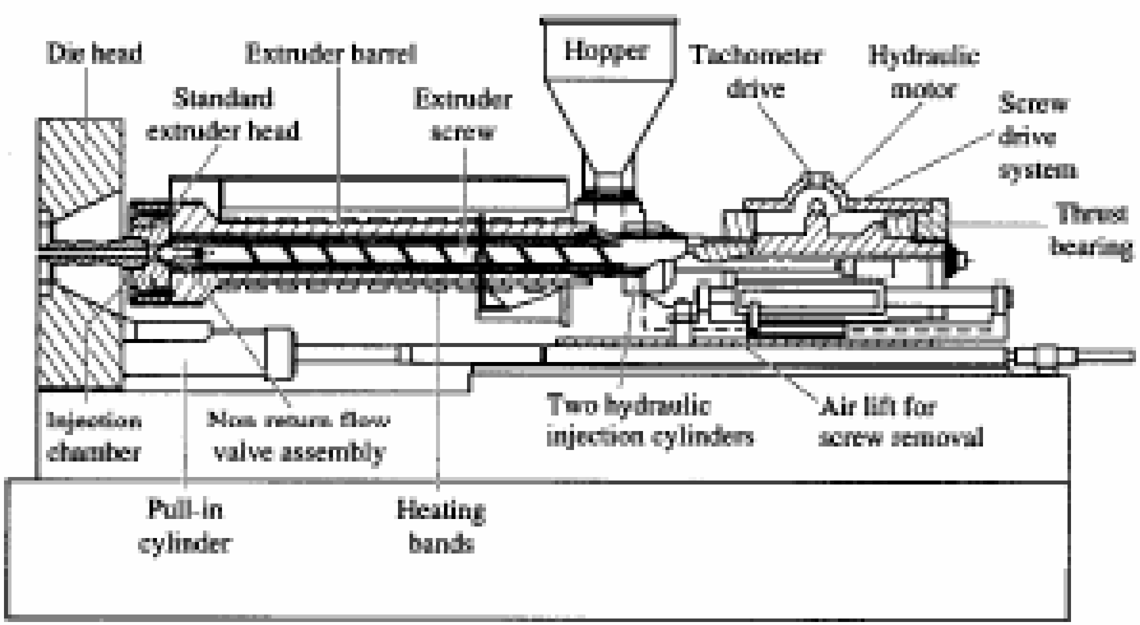
•Classification of injection molding are:

•Reciprocating Screw machine:

- •The market is now dominated by reciprocating screw type of injection moulding machine.
- • Injection and plasticizing are accomplished in the same screw and barrel.
- •An extruder type screw in a heated barrel performs a dual role. On the one hand it rotates in the normal way to transport, melt and pressurize the material in the barrel but it is also capable, whilst not rotating, of moving forward like a plunger to inject melt into the mold.

• Most energy for melting plastic comes from mechanical shearing.

• Plasticing behavior is similar to that of an extruder. i.e. The screw of injection molding is similar to that of an extruder.



INJECTION MOLDING

•Injection Molding Cycle Process:

(a) After the mould closes, the screw (not rotating) pushes forward to inject melt into the cooled mould. The air inside the mould will be pushed out through small vents at the furthest extremities of the melt flow path.

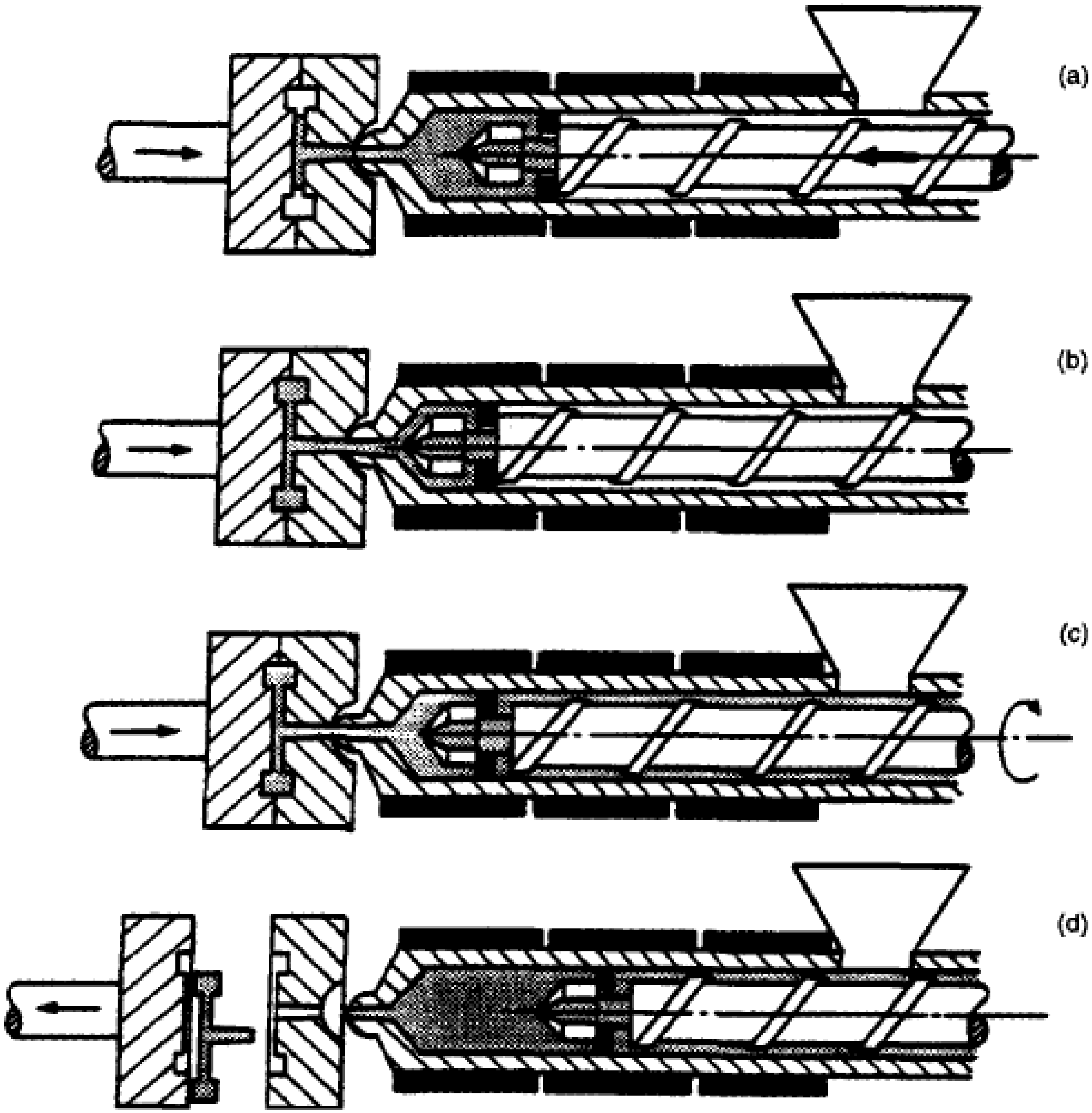
(b) When the cavity is filled, the screw continues to push forward to apply a holding pressure. This has the effect of squeezing extra melt into the cavity to compensate for the shrinkage of the plastic as it cools. This holding pressure is only effective as long as the gate remain open.

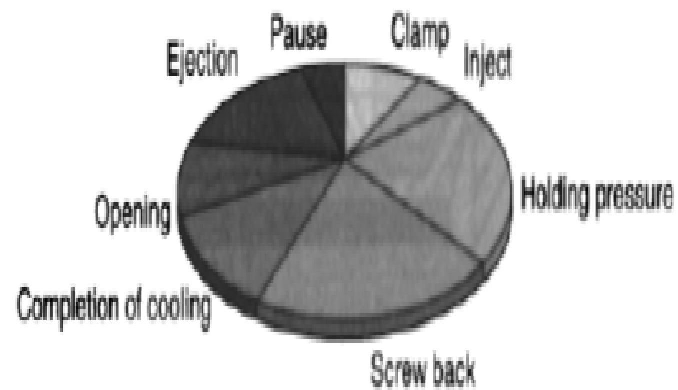
(c) Once the gate(s) freeze, no more melt can enter the mould and so the screw-back commences. At this stage the screw starts to rotate and draw in new plastic from the hopper. This is conveyed to the front of the screw but as the mould cavity is filled with plastic, the effect is to push the screw backwards. This prepares the next shot by accumulating the desired amount of plastic in front of the screw. At a pre-set point in time, the screw stops rotating and the machine sits waiting for the solidification of the moulding and runner system to be completed.

(d) When the moulding has cooled to a temperature where it is solid enough to retain its shape, the mould opens and the moulding is ejected. The mould then closes and the cycle is repeated.

INJECTION MOLDING

•Injection Molding Cycle Process:





INJECTION MOLDING

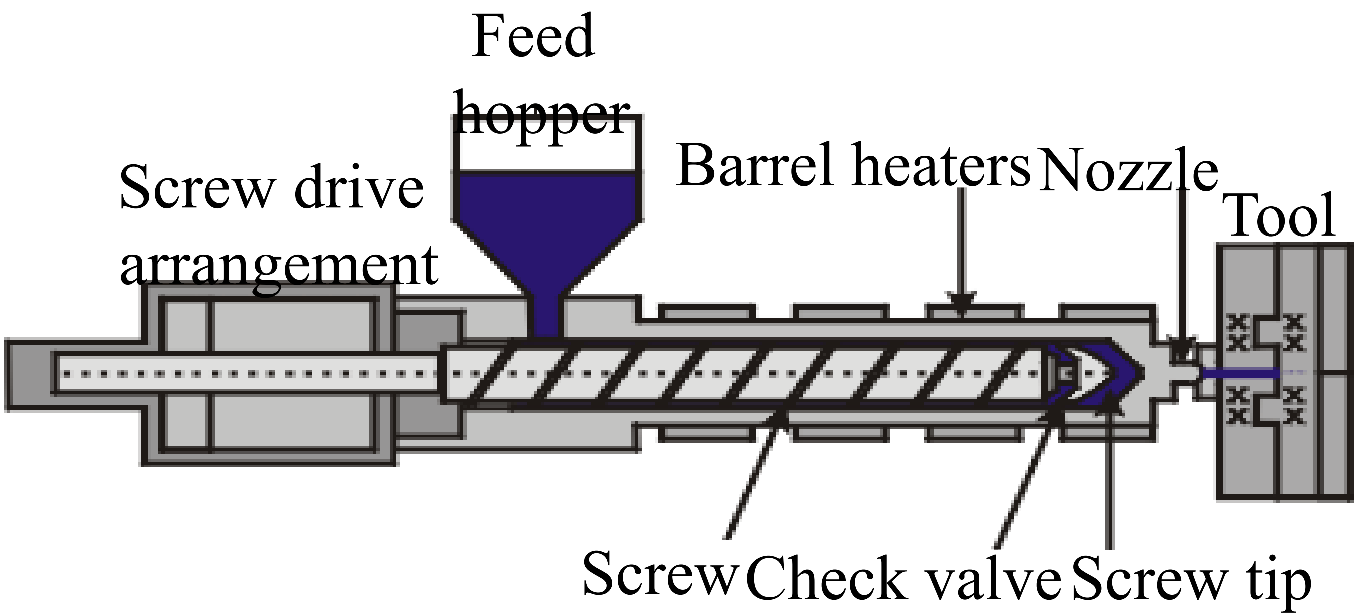
•**Injection Molding Machine Construction:**

Feed Hopper, Feed Throat and Barrel:

•Feed hopper feeds granular material to the screw.

•Feed throat is the section where material is introduced into the screw channel. It fits around the first few flights of the screw. Smoothest transition from feed hopper to feed throat will occur if the cross-sectional shape of the hopper is the same as the shape of the feed opening.

•Barrel is simply a flanged cylinder. It has to withstand high pressures & should posses good structural rigidity to minimize sagging or deflection. Barrels are made with wear resistant inner surface to increase the service life.



INJECTION MOLDING

•**Injection Molding Machine Construction:**

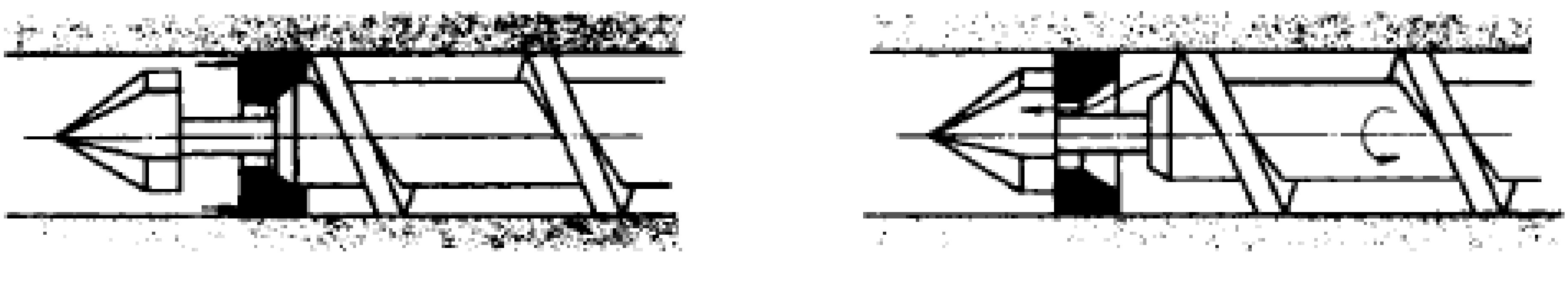
Screw:

•The screws used in these machines are basically the same as those described earlier for extrusion.

•The compression ratios are usually in the range 2.5:1 to 4:l and the most common L/D ratios are in the range 15 to 20.

•One important difference from an extruder screw is the presence of a back-flow check valve at the end of the screw.

•The purpose of this valve is to stop any back flow across the flights of the screw when it is acting as a plunger.



(a) Valve closed (b) Valve open

INJECTION MOLDING

•**Injection Molding Machine Construction – CLAMPING SYSTEM:**

•Two most common methods of clamping are used: Mechanical toggle system and hydraulic clamping.

•The purpose of clamping is to hold two or more parts of molds together.

•In order to keep the mould halves tightly closed when the melt is being injected under high pressures it is necessary to have a clamping system.

•Clamping force is normally rated in tons. 5, 15, 50, ... 7,000 tons.

•<u>MECHANICAL TOGGLE SYSTEM</u>:

•Mechanical units are toggle type, actuated by a lead screw or hydraulic cylinder. Single & double toggle system.

Advantages:

1. Economical in both equipment and operating
2. Speed of operation, toggle mechanism is fast
3. Self-locking, once the links have reached their extended positions, they will remain there until retracted.

Disadvantages:

1. The clamping force may not remain constant, as the temperatures the mold and tie bar change, their length changes.
2. Difficult to control the speed and force of the toggle mechanism
3. The system is susceptible to wear.

INJECTION MOLDING

•Injection Molding Machine Construction – CLAMPING SYSTEM:

Hydraulic system: In the hydraulic system, oil under pressure is introduced behind a piston connected to the moving platen of the machine. This causes the mould to close and the clamp force can be adjusted so that there is no leakage of molten plastic from the mould.

Advantages

- The clamping force can be monitored by a pressure gauge on the hydraulic system
- The clamping force can be changed at any time with hydraulic controls
- The stroke of the ram can be set
- The platen accelerates and de-accelerates smoothly
- Wear is less compared with the toggle system

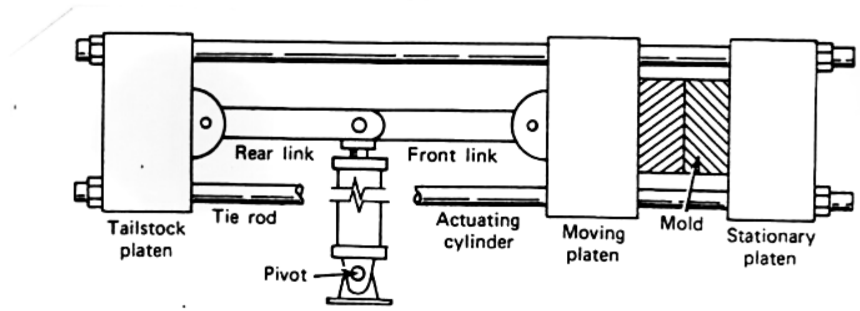
Disadvantages

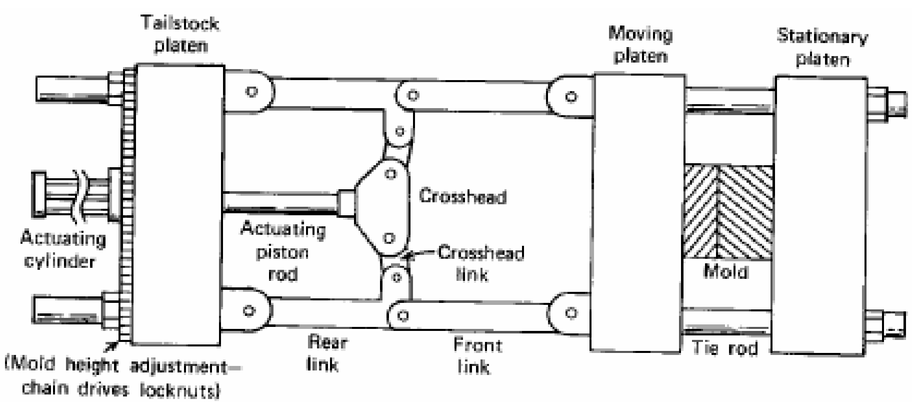
- Higher initial cost, and • Higher operating cost.

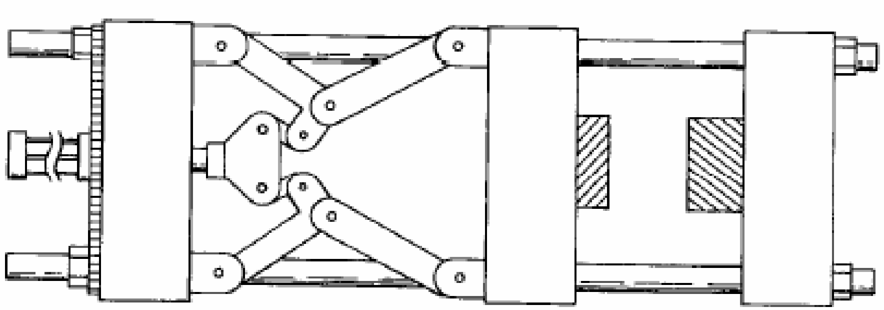
INJECTION MOLDING

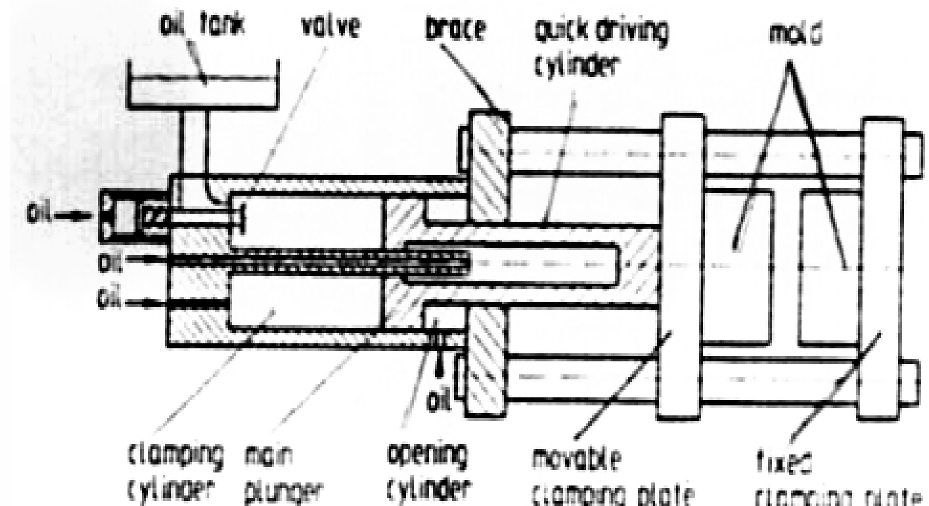
•Injection Molding Machine Construction – CLAMPING SYSTEM:

Mechanical and Hydraulic system:









INJECTION MOLDING

•**Injection Molding Machine Construction – CLAMPING SYSTEM:**

Difference between Mechanical and Hydraulic system:

HYDRAULIC	TOGGLE
Much higher original cost	Lower maintenance cost
Higher power needed, therefore more expensive to run	Lower power
Non positive clamp	Positive clamp
Unlimited stroke potential	Limited stroke
Direct reat out of clamp force	No direct readout
Ease adjustment of clamp force	Clamp force adjustment difficult
Easy mould set up	More involved mould setup
Varies stroke to mold height	Constant mould stroke
Easily controlled clamp speed	Clamp speed more difficult to control and stop
Lower maintenance & self lubricated	Higher maintenance cost

INJECTION MOLDING

- **Injection Molding Machine Ratings:**
- Shot Capacity: Maximum weight that can be injected into a mold cavities usually quoted in grams of PS.
- Plasticizing rate: Signifies the rate at which melt will be produced. (cc/s or kg/hr)
- Dry Cycle time: Time for hydraulic and electrical systems to run through an injection cycle, without actually injecting any melt into the mold.
- Injection Rate: Maximum rate at which the injection cylinder can eject fully plasticized material into the air on a single shot basis expressed in cc/s.
- Injection Pressure: Pressure on the plasticized material that the screw cylinder can transmit.
- Clamping stroke: The maximum distance the moving platen move.
- Minimum mold thickness: Closest distance that the two plate can come to each other.
- Maximum daylight: Furthest distance the plates can be separated from each other.
- Clearance between tie rods: Determines the size of the mold which can be used.
- Clamping tonnage: It is the force in tons, which can be applied to close the molds. It should be more than the maximum force developed due to injection of the plastic melts acting on the part of the mold. $F = P_{cavity} \times \text{Projected area}$
- Low clamping force will lead to flashing.

INJECTION MOLDING

•Effect of Material Properties on product quality:

While processing thermoplastic melts, the following factors should be taken into account in order to process efficiently and obtain quality products:

•Water absorption of raw materials: If plastic absorbs water, then dimensional changes will occur leads to reduction in mechanical properties as well as quality of the output will not be good. Small volume of water can generate steam which will be trapped within the compound during processing and will expand on decompression of the melt in latter stages which leads to voids in the finished product.

•Physical form of raw material: Granules of more regular shape and even size can lead to much higher throughput rates and much more even heating and hence better control in flow properties of polymer.

•Thermal stability of polymer: Polymers vary enormously in their thermal stability and it is worthy to consider the thermal stability characteristics of polymers. For eg., Polyacetals and PMMA depolymerise to monomer on heating. At processing temperatures such monomers are in the gaseous phase and even where there is only a small amount of depolymerisation a large number of bubbles can be formed in the products.

INJECTION MOLDING

- **Effect of Material Properties on product quality:**
- Flow properties of the molten plastics materials: Flow properties will generally concerned with the ability to fill the cavity. Greater the flow length ratio, then the mold will fill.
- Adhesion of melt to metal:
- Thermal properties affecting cooling of melt: It will influence production rates as well as properties of the product. Longer solidification time will lead to longer cycle time. After processing, the polymer melt will tend to re-coil or relax, if the mass solidifies before relaxation, the product will be anisotropic with respect to mechanical properties.
- Shrinkage:
- Orientation: To avoid distortion of product.

INJECTION MOLDING

- **Shrinkage:**
- In injection molding, plastic raw material is melted, subjected to very high pressures, cooled down to demolding temperature inside the mold and then cooled naturally to room temperatures.
- The final parts are always somewhat smaller than the mold cavity due to shrinkage effects.
- Shrinkage of injection molded products is affected by processing parameters and mold geometry.
- Amorphous plastics have low shrinkage than that of crystalline materials.
- Filled thermoplastics usually shrink less than that of non-filled materials.
- Non-uniform wall thickness, sharp angles and long flow paths can cause uneven shrinkage.
- Thermosetting plastics shrink much less than thermoplastics and to be considered if close dimensional tolerances are essential.
- Shrinkage in the flow direction will be greater than shrinkage in transverse direction, because of orientation.

INJECTION MOLDING

•**Factors affecting Shrinkage:**

•Effect of Injection Pressure: As injection pressure is increased, overall cavity pressure will be high and hence more packing will be possible and shrinkage will reduce.

•Effect of Melt Temperature: As melt temperature is increased, pressure transmission is significantly increased due to decrease in viscosity. The higher pressure overrides the effect of higher temperature.

•Effect of Mold Temperature: As the part continues to cool, increasing the thickness of the frozen layer. The thicker the frozen layer the lower the shrinkage. Hence higher mold temperatures give more shrinkage.

•Effect of Injection Rate: As injection rate is increased, efficiency of transmission of pressure increases, hence shrinkage reduces.

•Effect of gate dimension: As gate dimension increases, shrinkage reduces. It will take longer time for freezing, more time for packing, hence shrinkage is less.

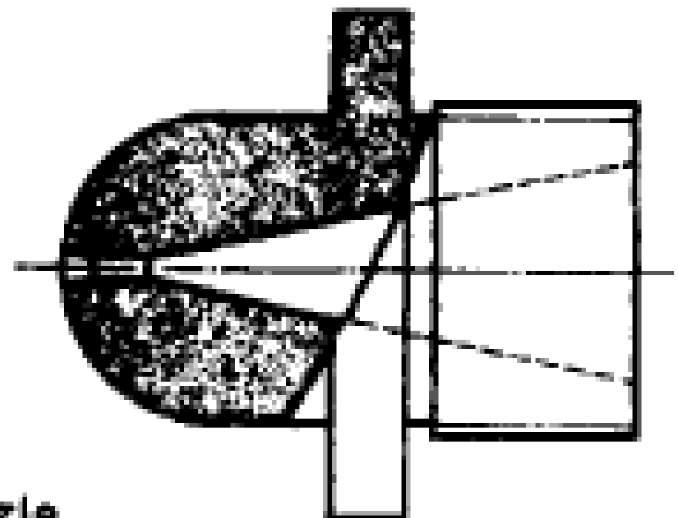
•Effect of part thickness: Thicker the part, higher is the shrinkage. The hot interior has more hot plastic to shrink per unit volume.

INJECTION MOLDING

•NOZZLE:

•It is screwed into the end of the barrel and provides the means by which the melt can leave the barrel and enter the mold.

•Contact with the cold mold causes heat transfer from the nozzle, hence withdrawing the nozzle from the mold during screw back, otherwise plastic may freeze off in the nozzle.



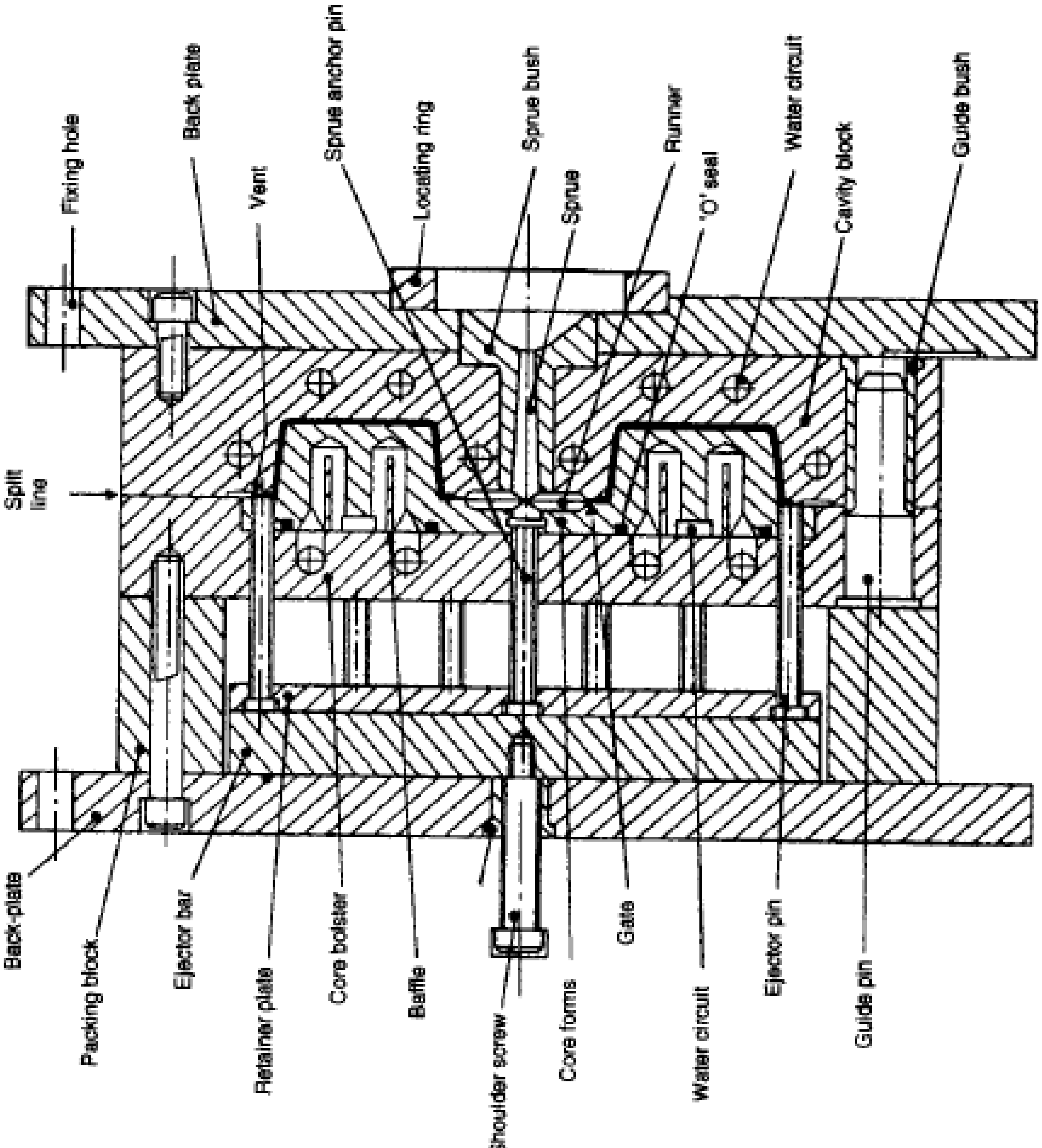
(a) Open nozzle

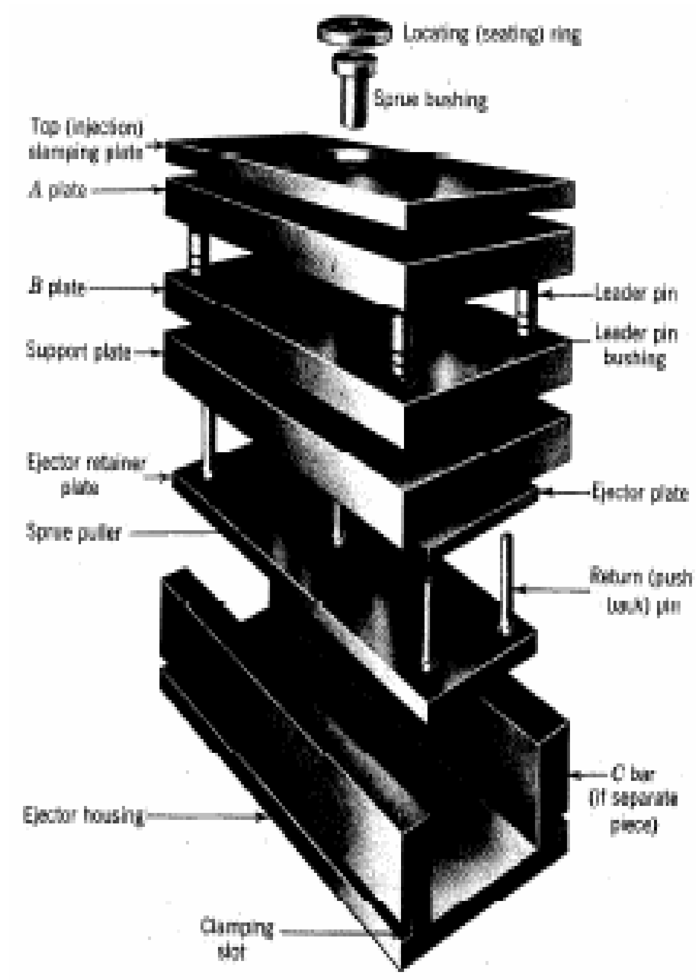
INJECTION MOLDING

- BASIC MOLD CONSTRUCTION:
- Basic function of a injection mold are accommodation and distribution of the melt, shaping and cooling of the material, solidification of the melt and ejection of the molding.
- STEPS FOR MOLD DESIGN:
  - Part Geometry, material, tolerance & machine data
  - No of cavities
  - Cavity layout
  - No of Parting line
  - Dimensions of cavity
  - Sprue and Runner
  - Gate
  - Cooling system
  - Mechanical design
  - Ejector system
  - Alignment system
  - Venting
  - Mounting option

INJECTION MOLDING

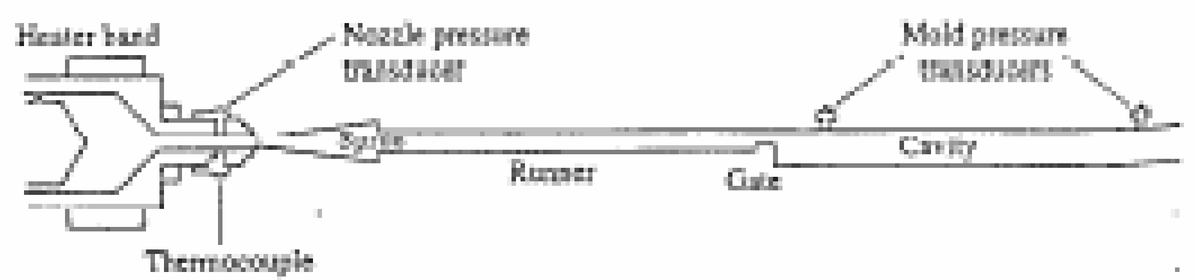
- BASIC MOLD ELEMENTS:





INJECTION MOLDING

- BASIC MOLD ELEMENTS:
- Locating Ring: It is used to locate the centre of the nozzle to the centre of the mold.
- Sprue bush: It is fitted with the locating ring tightly and the melt enters through this sprue bush from the nozzle.
- Cavity & core plate: This plates are used to give the final shape of the molding.
- Runner & Gate: The melt from the sprue passes through the runner & gate and enters in to the cavity area. It is fitted in the cavity & core plate.
- Ejection system: It is used to eject the component from the mold.
- Push back pin and push back rest pin: It is used to bring back the ejection system to its normal position after ejection.
- Knock out rod: It is used to move the ejection system forward.
- Guide pillar & guide bushes: It is used for the alignment of mold halves perfectly after opening & closing.



INJECTION MOLDING

•BASIC MOLD ELEMENTS:

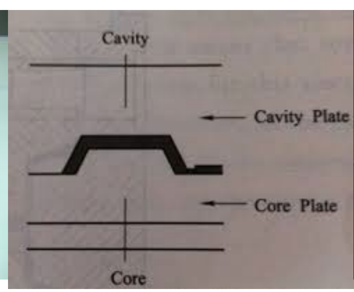
•Core back plate: It is used to give additional support for the core plate to withstand higher pressures.

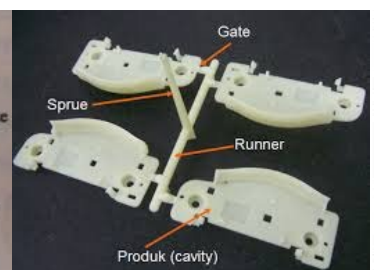
•Spacer block: It is used to fix the ejection system in the mold.

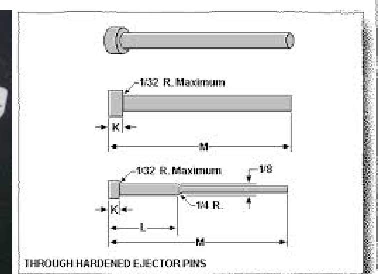
•Socket head screw: It is used to fix the different mold plates together.

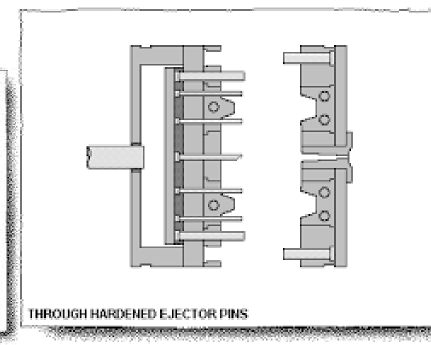
•Fixed half and moving half: The plates in the fixed half are Cavity retainer plate & cavity plate. The plates in the moving half are core plate, core back plate, spacer block and core retainer plate. All plates of each half are fixed together with a socket head screw.

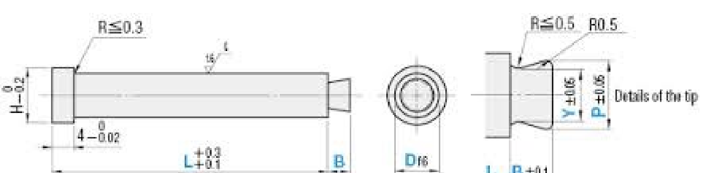
•Sprue puller: It is used to pull off the sprue from the mold.

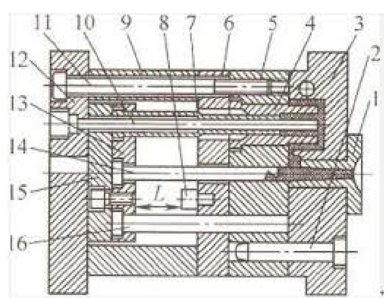












INJECTION MOLDING

•CLASSIFICATION:

•There are many different types of molds, designed to meet many different product requirements.

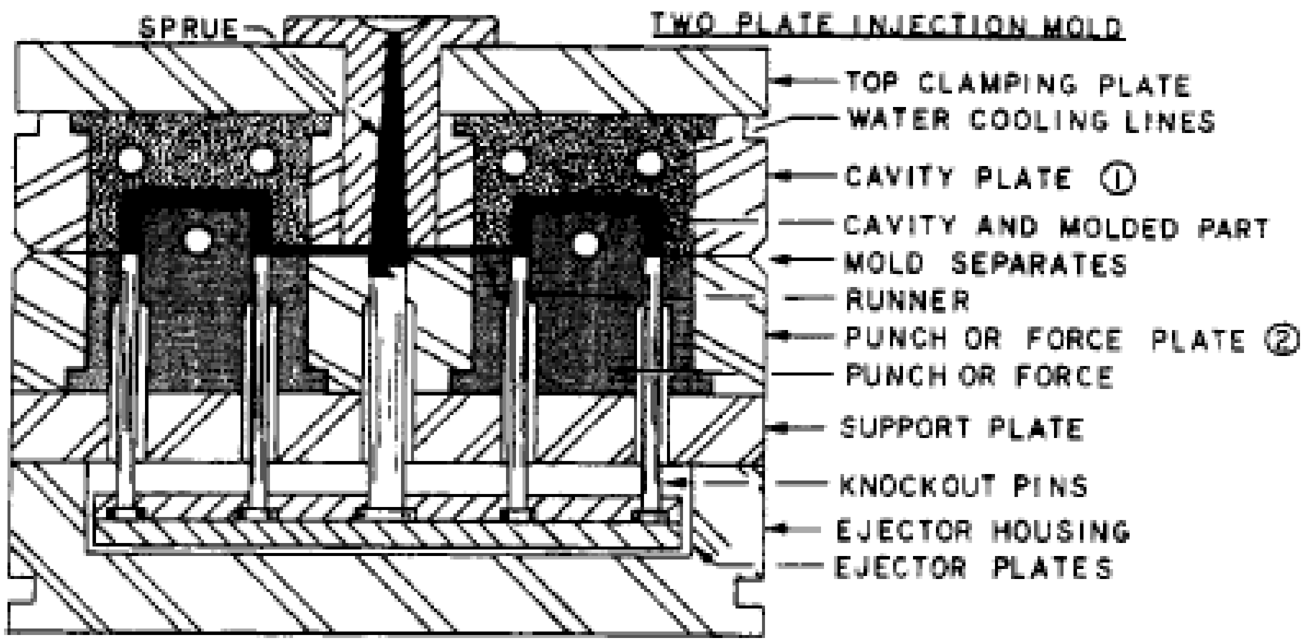
•Industries generally identifies some basic types of molds for use with thermoplastic materials.

•TWO PLATE MOLD:

•It consists of two plates with cavity and core mounted in either plate.

•Plates are fastened to the platens of fixed and moving half.

•Basic design of injection molds have this design concept.



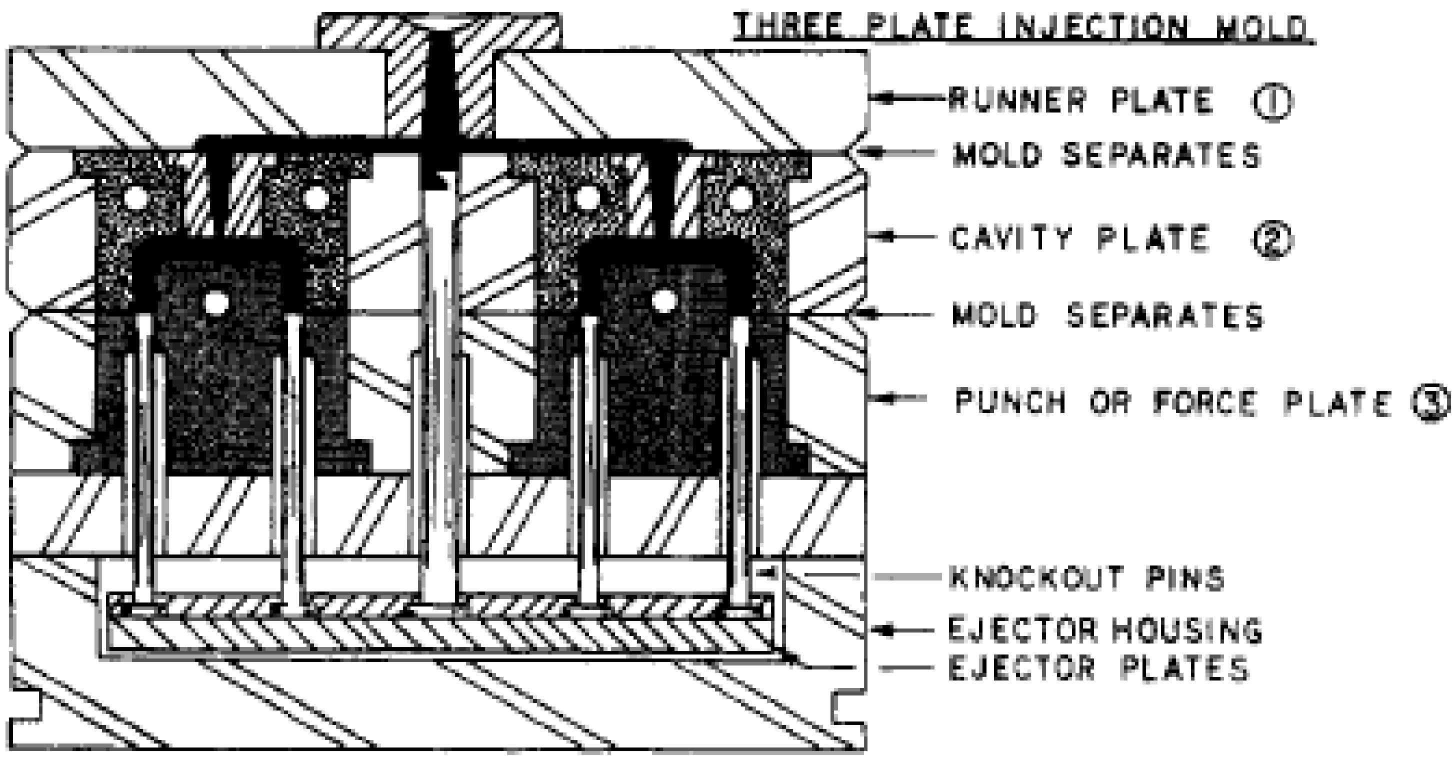
INJECTION MOLDING

•CLASSIFICATION:

•THREE PLATE MOLD:

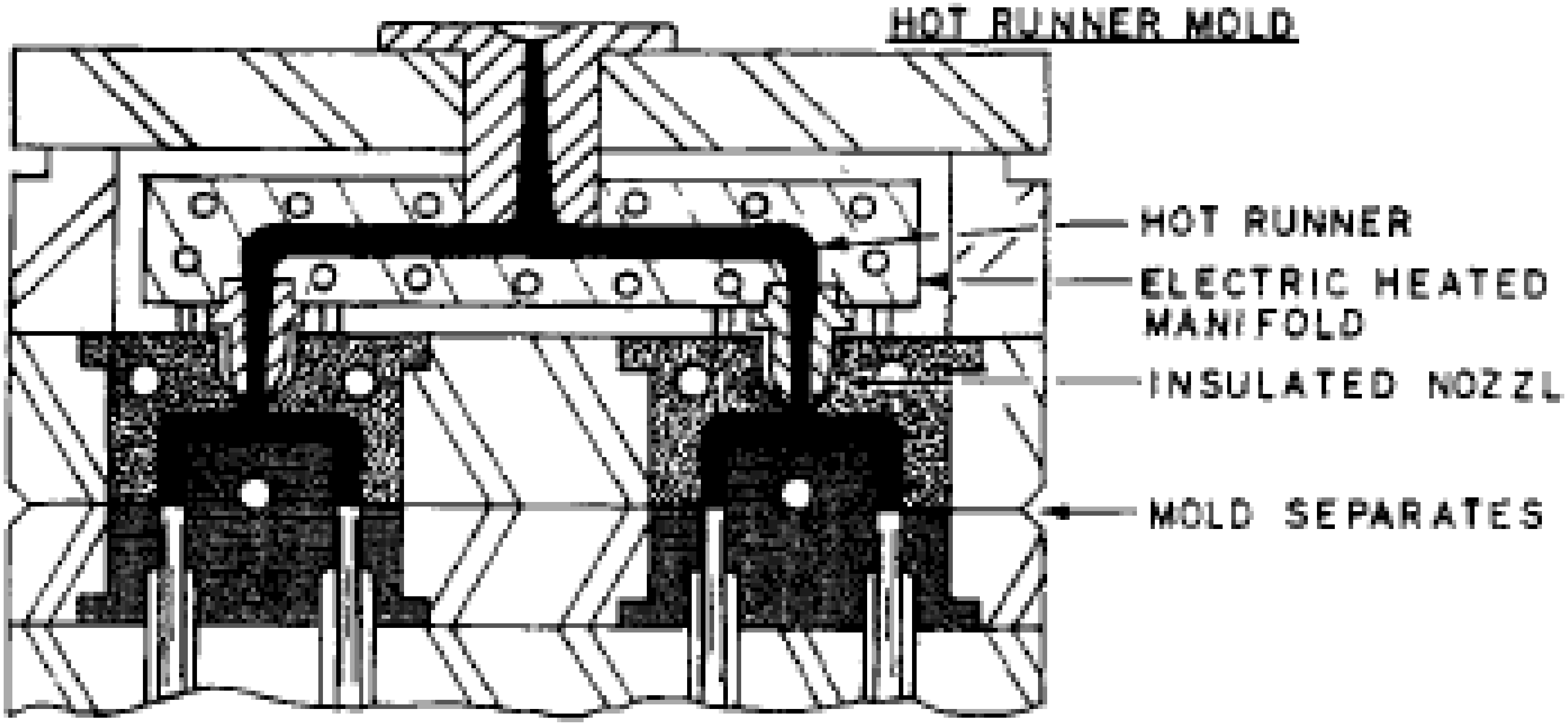
•It is made up of three plates: Runner plate, which is attached to the stationary platen and usually contains the sprue and half of the runner; (2) middle or cavity plate, which contains half of the runner and gate and is allowed to float when the mold is open; (3) movable plate which contains the molded part and ejector system.

•It is used to segregate the runner system and the part when the mold opens.



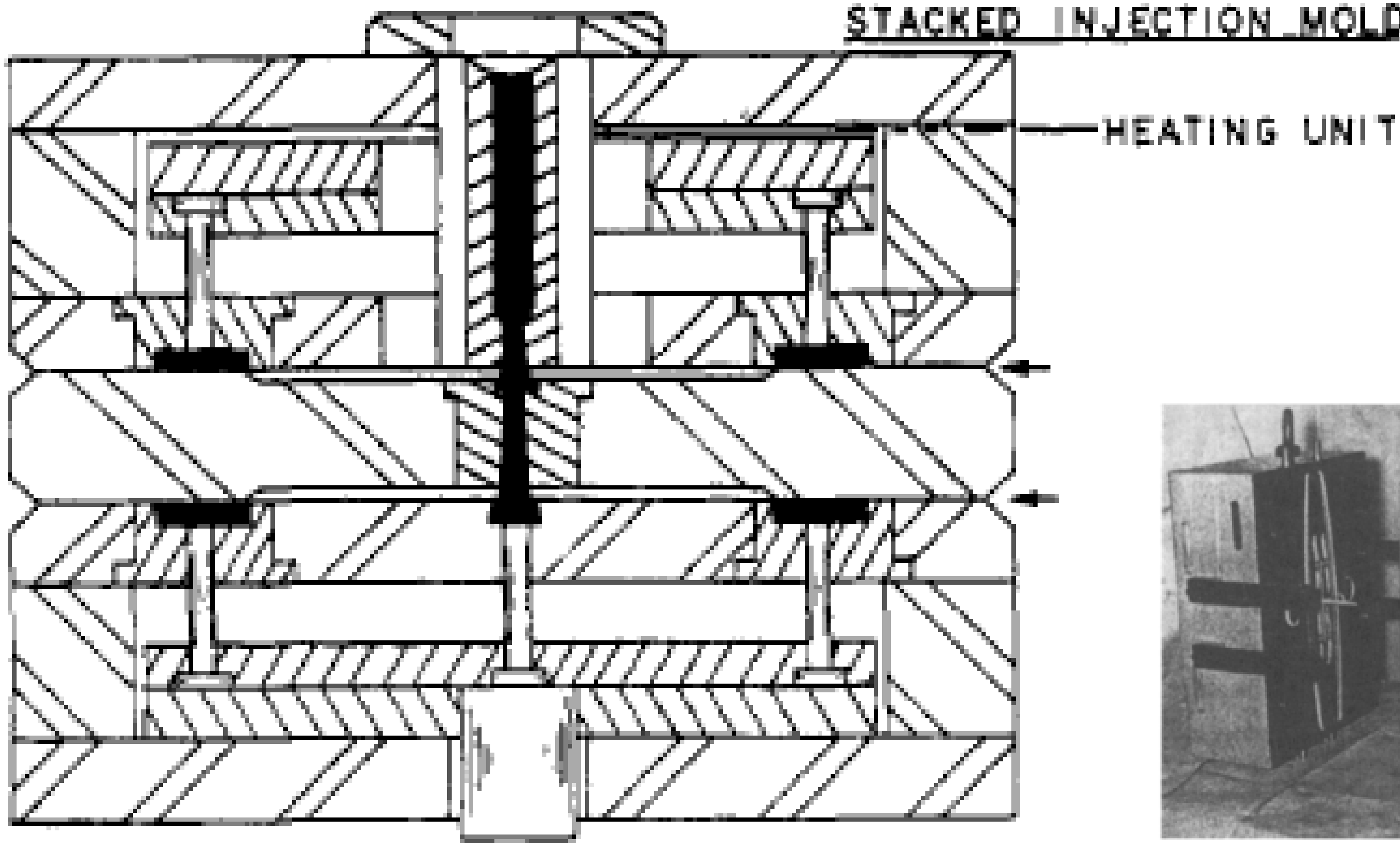
INJECTION MOLDING

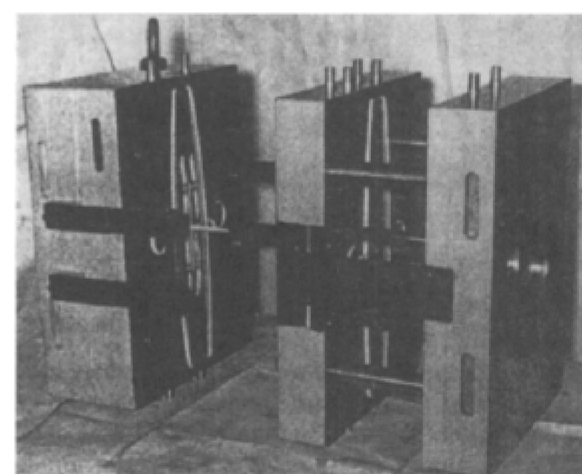
- CLASSIFICATION:
- HOT RUNNER MOLD:
- In this type of mold, the runners are kept hot in order to keep the molten plastic in a fluid state at all times.
- It is also called runnerless molding process.
- The runner is contained in a plate of its own and is similar to three plate mold, except the runner section of the mold is not open during molding cycle.



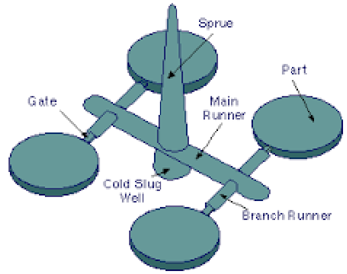
INJECTION MOLDING

- CLASSIFICATION:
- STACK MOLD:
- Basically, a stacked mold is a multiple two plate mold, with the molds placed one on top of the other.
- It doubles the output from a single press.





INJECTION MOLDING



•FEED SYSTEM:

•It consists of SPRUE, RUNNER (Primary & Secondary) & Gate

•SPRUE:

•It receives the plasticized material from the nozzle of the barrel & guides it to the plane through the parting line which is usually perpendicular to the sprue.

•The material flows directly from the nozzle into the sprue. This procedure puts a very high load upon the mold & it wears out fast. So employ of sprue bush is required, made of hardened steel & inserted into the mold. It can be replaced if damaged or worn out.

•Internal aperture of the sprue bush has between 2 – 4 deg included taper to remove sprue from the mold at the end of molding cycle.

•Two basic design of sprue bush which differs w.r.t the form of seating: (i) Spherical seating & (ii) flat seating.

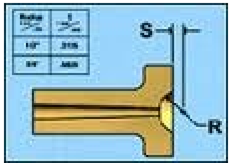
•Spherical recess is used for spherical ended nozzle while flat seating is used for flat ended nozzle.

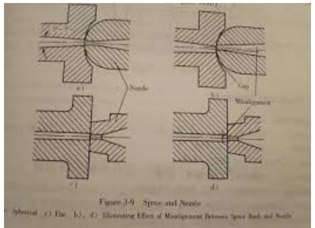
•Radius of nozzle is slightly less than in bush.

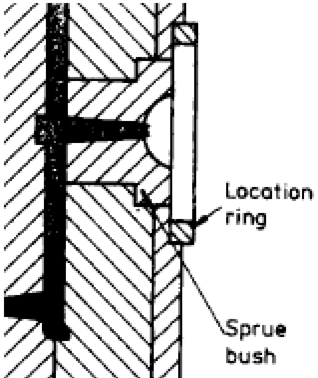
•Dimensions of sprue depend primarily on the dimensions of molding & thickness. Factors to be considered are:

Sprue must not freeze before any other cross-section.

Sprue has to be demold easily.







INJECTION MOLDING

•FEED SYSTEM:

•It consists of SPRUE, RUNNER (Primary & Secondary) & Gate

•RUNNER:

•Runners connect the sprue via gate with the cavity.

•Standard runners are directly machined into the mold plates.

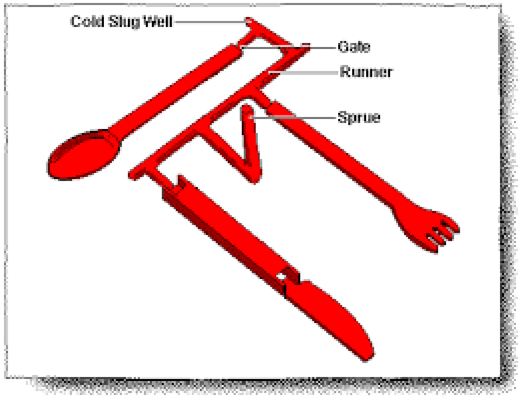
•Distribute the material in such a way that melt of the same state & same pressure fills all cavities at the same time.

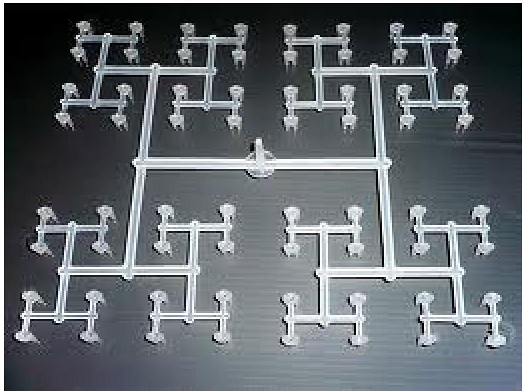
•CONSIDERATIONS FOR A DESIGNER:

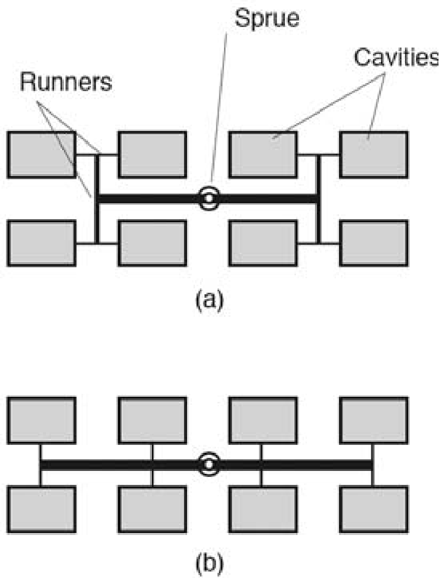
•Shape of the cross-section of the runner

•Size of the runner

•Runner layout







INJECTION MOLDING

- FEED SYSTEM:
- It consists of SPRUE, RUNNER (Primary & Secondary) & Gate
- SHAPE OF THE CROSS-SECTION OF RUNNER:
- Cross sectional shape of runner is usually one the given forms:
- Fully round   trapezoidal   Modified trapezoidal   hexagonal
- Runner should provide maximum cross-sectional area for pressure transfer and minimum contact area on the periphery for heat transfer.
- Ratio of cross-sectional area to periphery will indicate the efficiency of runner design.
- Higher the value the greater the efficiency.
- Round and square type of runner are the two most satisfactory design.

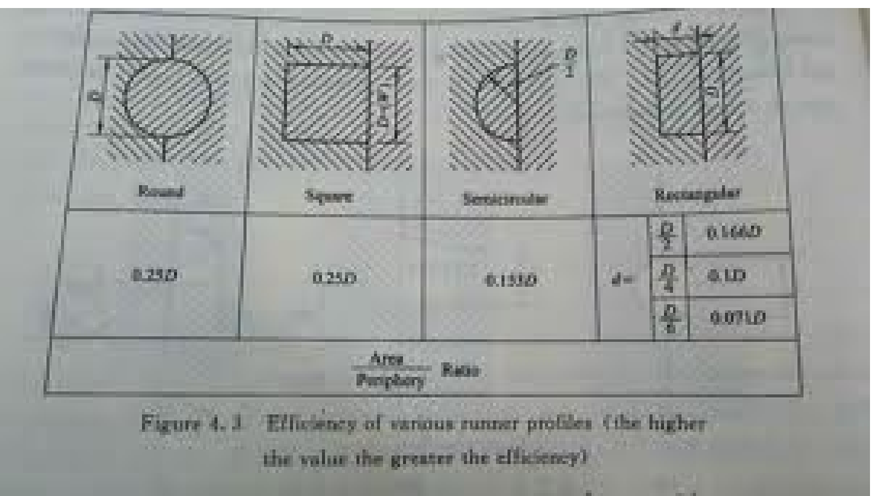


Figure 4.3 Efficiency of various runner profiles (the higher the value the greater the efficiency)

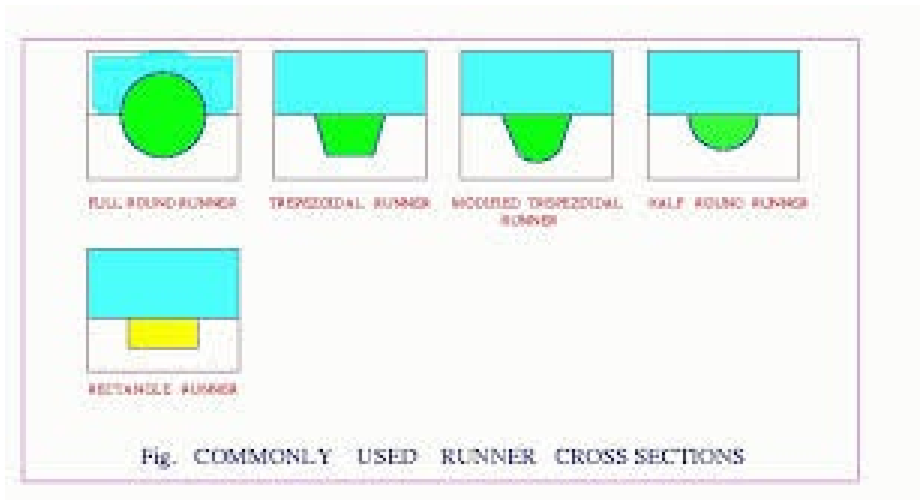


Fig. COMMONLY USED RUNNER CROSS SECTIONS

INJECTION MOLDING

- FEED SYSTEM:
- It consists of SPRUE, RUNNER (Primary & Secondary) & Gate
- SHAPE OF THE CROSS-SECTION OF RUNNER:
- Square runner is not very satisfactory due to difficult to eject.
- In practice, angle of 10 deg is incorporated on the runner wall, thus becomes trapezoidal section.
- Volume of trapezoidal runner is approximately 25% greater than the round runner. To reduce this difference, a modified trapezoidal form has been developed which only 14% greater than round runner.
- Hexagonal runner basically a double trapezoidal runner. The cross-sectional area of this runner is about 82% of round runner.
- It is easy to match two halves of hexagonal runner compared to matching round runner.
- For simple two plate mold which have a flat parting surface the fully round or hexagonal runner is to be preferred.
- For complex parting surface, trapezoidal or modified trapezoidal runners are preferred.

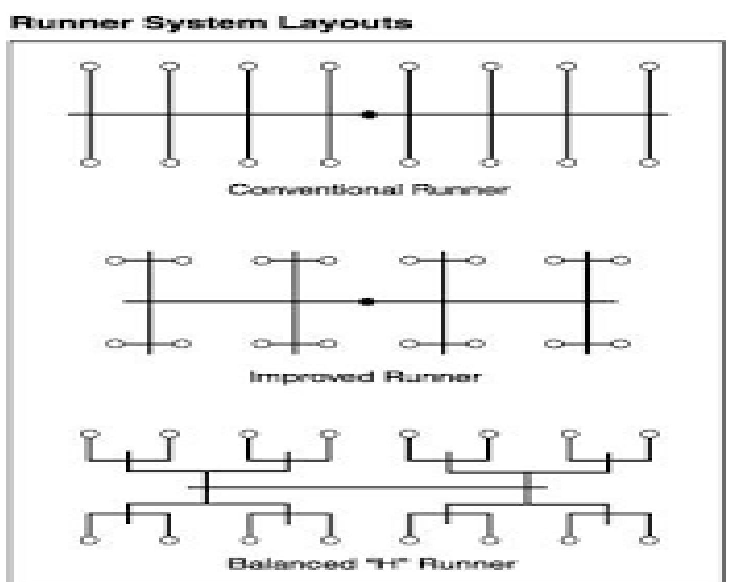
INJECTION MOLDING

- FEED SYSTEM:
- It consists of SPRUE, RUNNER (Primary & Secondary) & Gate
- RUNNER SIZE:
- Factors to be considered are:
- Wall section & Volume of molding
- Distance of the impression from the sprue
- Runner cooling consideration
- Range of mold making cutters available
- Plastic material to be used.

- Basically runner should not be below 2 mm diameter or above 10 mm diameter.
- Theoretically, the cross-sectional area of the main runner should be equal to or excess of the combined cross-sectional area of the branch runners.

INJECTION MOLDING

- •FEED SYSTEM:
- •It consists of SPRUE, RUNNER (Primary & Secondary) & Gate
- •RUNNER LAYOUT:
- •Factors to be considered are:
- •Number of impressions
- •Shape of the component
- •Type of mold
- •Type of gate
- •Finally runner length to be kept minimum to reduce pressure losses and should be balanced.



INJECTION MOLDING

- FEED SYSTEM:
- It consists of SPRUE, RUNNER (Primary & Secondary) & Gate
- GATE:
- It connects the sprue, runner to the impression.
- It has small cross-sectional area compared with rest of feed system.
- It must be located in such a way that rapid and uniform mold filling is ensured.
- FACTORS OF GATE:
- Gate freezes soon after the impression is filled and sealed off the runner system which prevents material being sucked out of the cavity during screw-back.
- Simple degating due to small cross-sectional area.
- Small witness marks remains.
- Gate must be located that air present in the mold cavity can escape during injection, Otherwise it leads to not fulfilled, either short or burnt spots on the molding.
- Location of gate must be such that weld lines are avoided. It reduce the strength and spoil the appearance of molding.

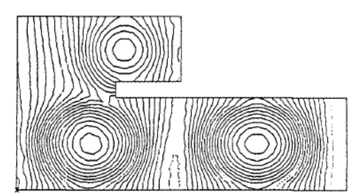
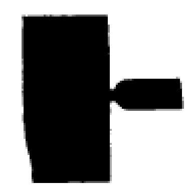


Fig. 4-54 Multiple-gate flow pattern.



(c) Side gate

INJECTION MOLDING

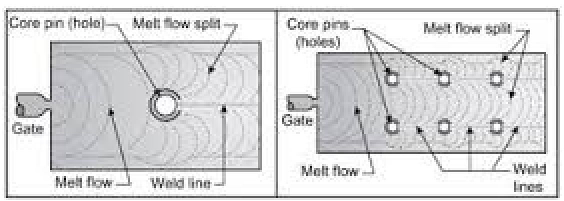
- FEED SYSTEM:
- It consists of SPRUE, RUNNER (Primary & Secondary) & Gate
- GATE:
- Factors depend on Size of gate in terms of gate cross-sectional area and gate length:
- Flow characteristics of the material
- Wall section of the molding
- Volume of material to be injected
- Temperature of the melt
- Temperature of the mold.
- TYPES OF GATE:
- Standard Gate
- Tab gate
- Fan gate
- Pinpoint gate

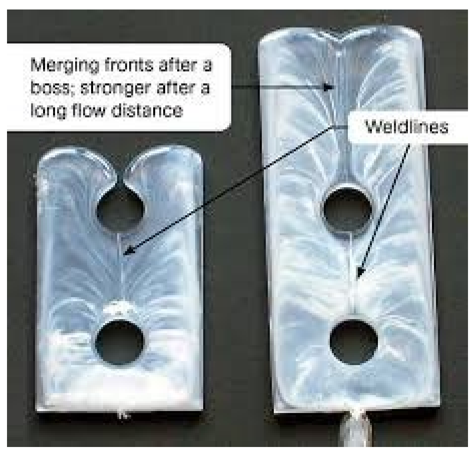
Ring gate
Disc gate
Sprue gate

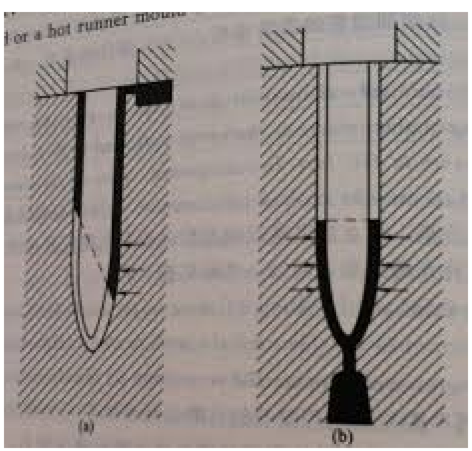
Submarine Gate
Film gate
Overlap gate

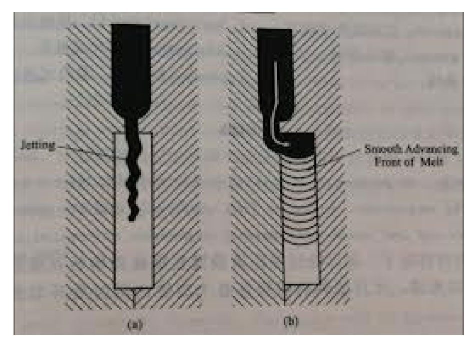
INJECTION MOLDING

- FEED SYSTEM:
- It consists of SPRUE, RUNNER (Primary & Secondary) & Gate
- POSITIONING OF GATE:
- Position of gate should be such that even flow of melt into the impression.
- Take care of two or more advancing fronts would rarely meet to form a weld line.
- No theoretical size exists for the ideal gate. Gate size for a particular component is normally based on past experience.









INJECTION MOLDING

•EJECTION SYSTEM:

•All thermoplastic materials contract as they solidify, that means that the molding will shrink on the core and this makes the molding difficult to remove.

•In normal practice, provide some means by which the molded part can be positively ejected from the core.

•The ejector system consists of ejector plate assembly and method of ejection

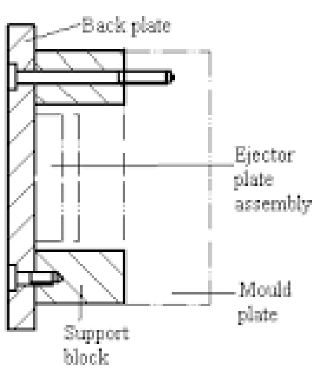
•EJECTOR PLATE ASSEMBLY:

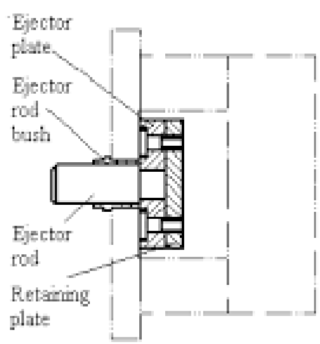
•It is the part of the mold to which the ejector element is attached.

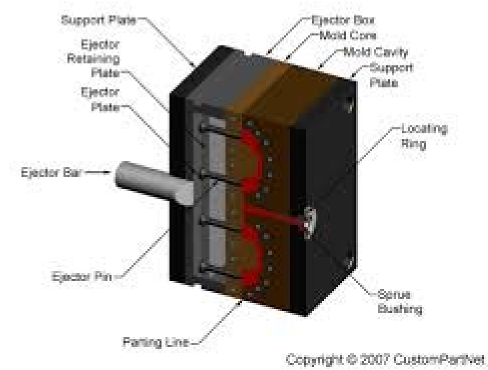
•It consists of ejector plate, retaining plate and an ejector rod.

•Ejector rod functions not only as a actuating member but also as a method of guiding the assembly. It is coupled directly to the hydraulic ram.

•Ejector plate is used to transmit the ejector force from the actuating system of the injection machine to the molding via an ejector element.







INJECTION MOLDING

•EJECTION SYSTEM:

•Designer has several ejection techniques but the choice will be restricted depending upon the shape of the molding.

•Basic ejection techniques are

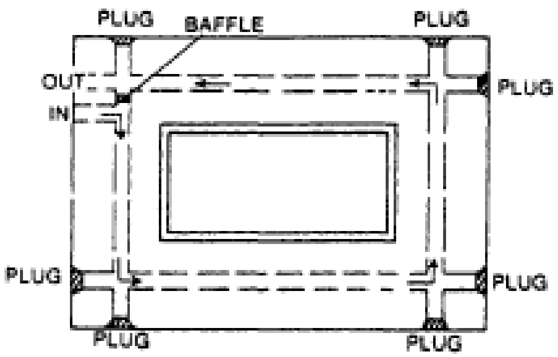
•Pin Ejection	Sleeve ejection	Bar ejection
•Blade ejection	Stripper plate ejection	

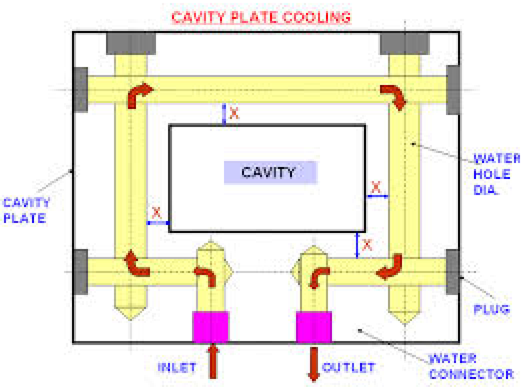
INJECTION MOLDING

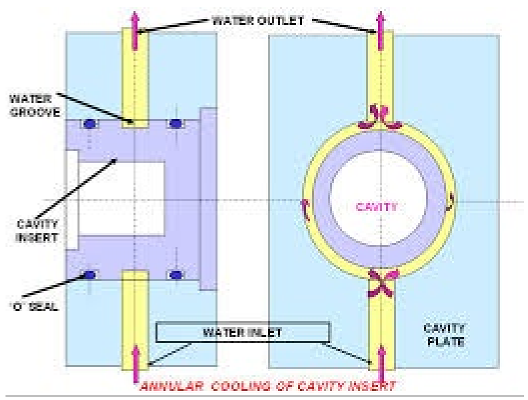
- COOLING SYSTEM:
- Temperature of mold is important as it governs a portion of the overall molding cycle.
- To maintain required temperature, water or other fluid is circulated through holes or channels within the mold.
- These holes or channels are termed as flow-ways and the complete system of flow ways is termed as cooling system.
- During impression filling, the hottest material will be at the entry point and the coolest will be at the farthest from the entry.
- When using drillings for the circulation of coolant, the circuits must not be positioned too close to the impression. (closer than 16 mm)
- The holes are normally interconnected to form a circuit.

INJECTION MOLDING

- •COOLING SYSTEM:
- •CAVITY PLATE COOLING:
- •Simplest circuit, Rectangular circuit and U circuit.

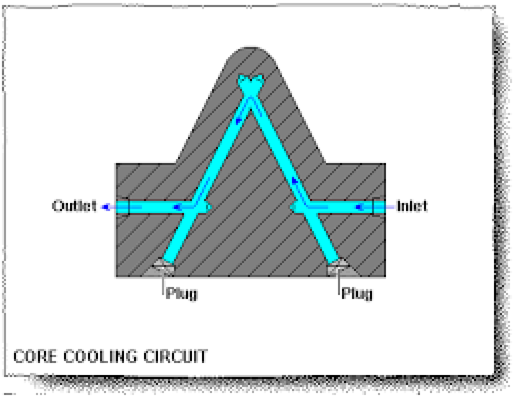


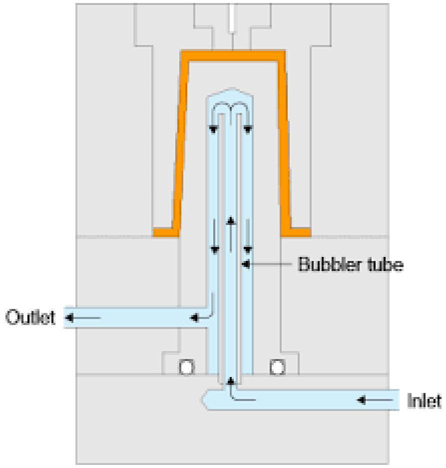


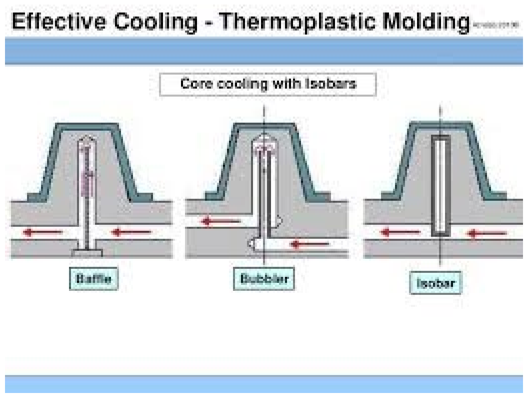


INJECTION MOLDING

- COOLING SYSTEM:
- CORE PLATE COOLING:
- Angled hole system, baffled straight hole and bubble system.
- Flow ways are drilled at an angle of the core plate.







SPECIALIZED INJECTION MOLDING

•**MICRO-INJECTION MOLDING**:

•It is used for molding miniature products with dimensional accuracy ranging from few mm to micrometer.

•Products weighing few mg to less than gms eg. Micro gear wheels, etc.,

• It has small plastication unit of 8 to 12 mm screw, L/D of 15 to avoid material degradation.

•Precise shot volume control and desirable injection rate.

•Increase mold wall temperature to avoid premature solidification of melt.

•Precise alignment and gently mold opening/closing.

•**Disadvantages:**

• 90% material wastage as over size runners are used.

•Higher temperature to avoid melt solidification leading to high cycle time.

•APPLICATION Products : Optical telecommunication, computer data storage, medical technology, Bio technology , camera components, read write heads of hard disc, Optical fibre switches and connectors, micropumps, high precision gears, medical sensors, micromotors, surgical instruments, telecommunication components.

SPECIALIZED INJECTION MOLDING

•**MULTI-COMPONENT INJECTION MOLDING**:

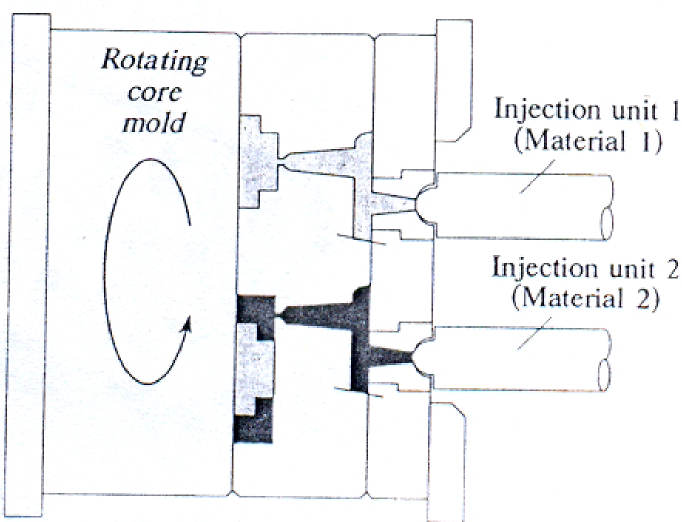
•It is used to make multicolor or multifunctional products at reduced cost.

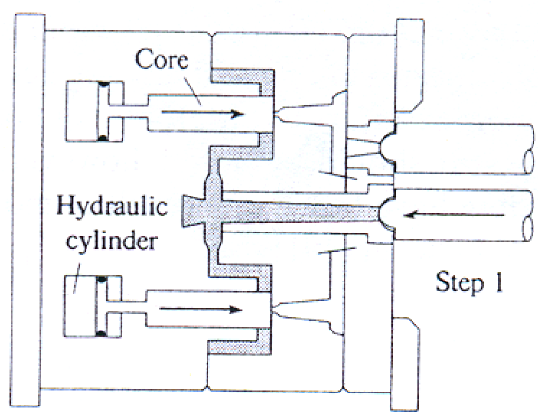
•This process injects a polymer over another plastic insert.

•In conventional approach, the insert can be molded in one operation and then transferred to a second mold, where it is overmolded with a second polymer.

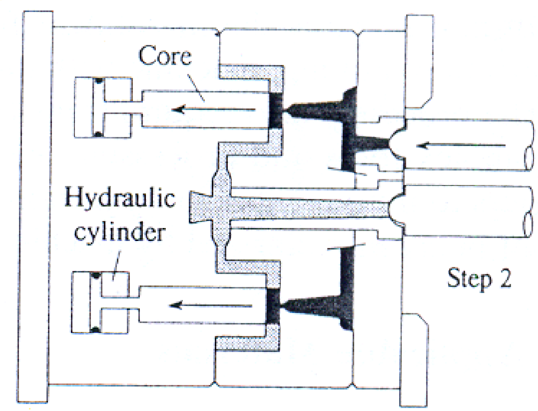
•Overmolding can be carried out by two methods: Rotating mold technique and Core pull technique.

•Products : Automotive: Multicolored tail light lenses, Air duct flam, Door rub strip and seal etc. Non Automotive : Type writer key, tooth brushes, tool handle, toys, TV remote control, etc.





First material is injected, cools and solidifies



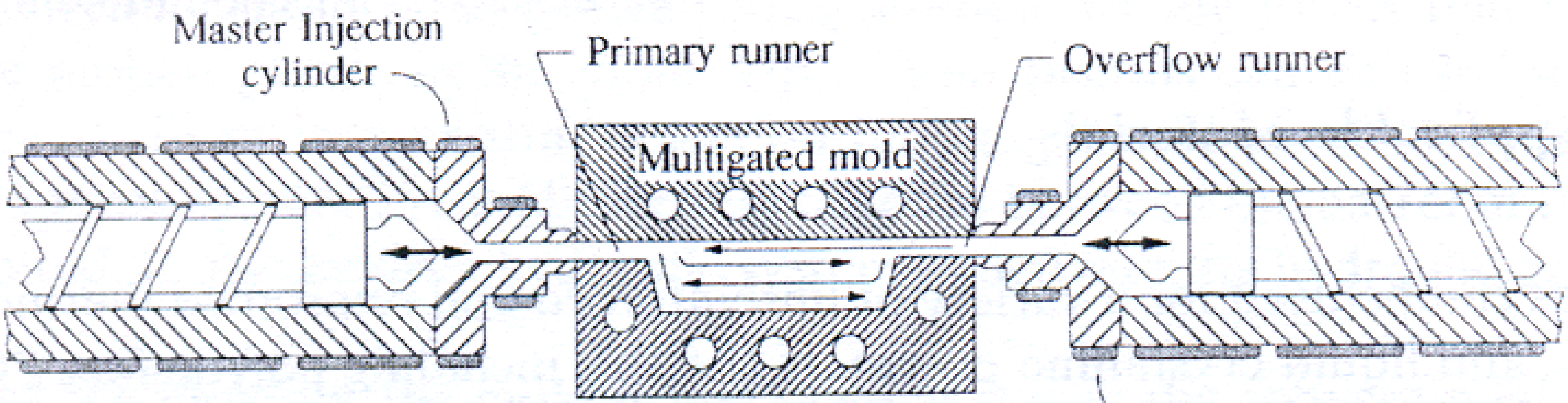
Core or slide retracts creating a void that is filled with the second material

SPECIALIZED INJECTION MOLDING

•**PUSH PULL INJECTION MOLDING**:

•It is a process in which the melt is pushed in and out of the mold between two injection units during injection and solidification stages.

•This induces molecular and fibre orientation along the direction of the movement.



The master screw overfills a multigated mold cavity. The melt flows to the companion screw and the process reverses a number of times.

SPECIALIZED INJECTION MOLDING

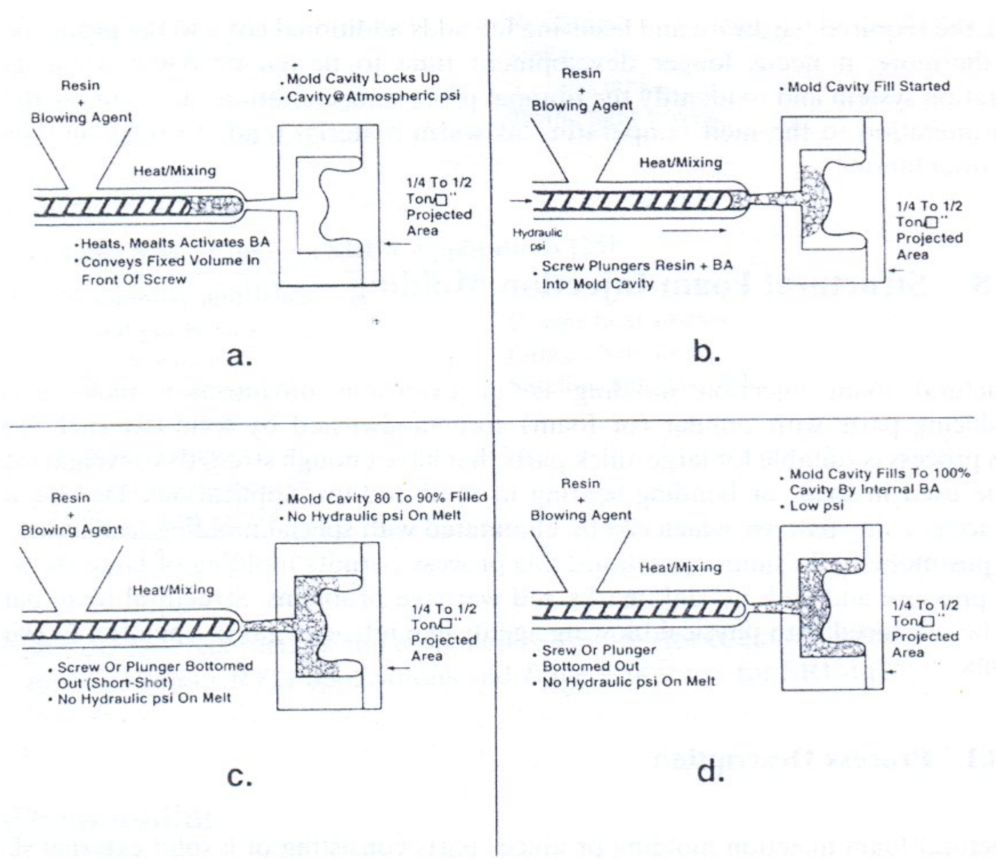
•**STRUCTURAL FOAM INJECTION MOLDING**:

•It produces parts of solid external skin surrounding an inner cellular foam core.

•This process is suitable for large thick parts with enough strength to weight ratio.

Gas: generally Nitrogen: the gas is dissolved in the melt in a accumulator before injection.

Chemical Blowing agent CBA: Azobizcarbonamide, AZ or Azo bisformamide ABFA,



SPECIALIZED INJECTION MOLDING

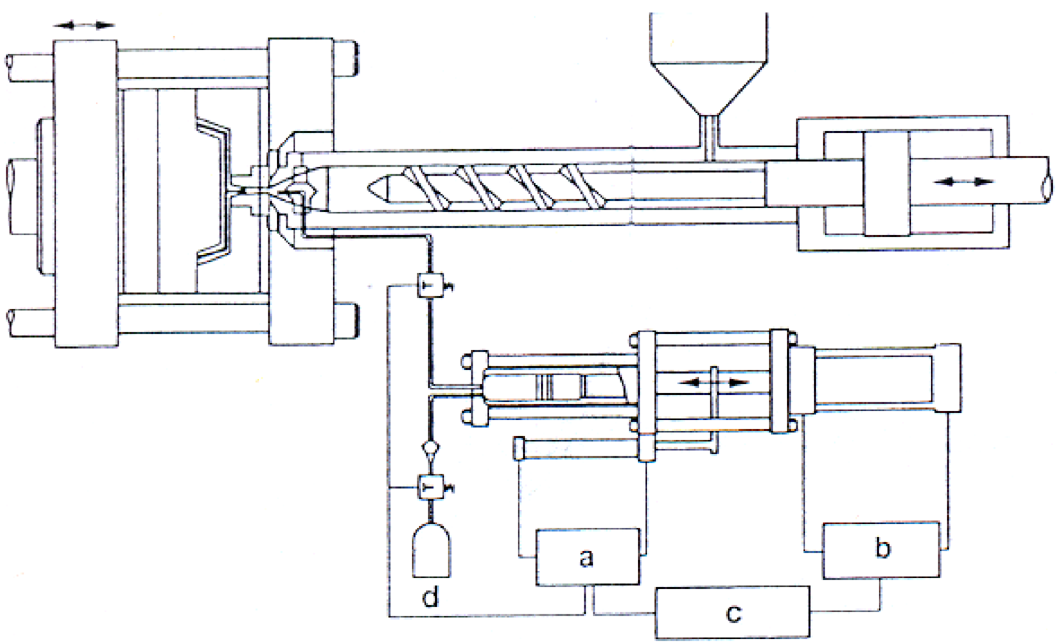
•**GAS-ASSISTED INJECTION MOLDING**:

•It consists of a partial or nearly full injection of polymer melt into the mold cavity, followed by injection of inner gas (typically nitrogen) into the core of the polymer melt through the nozzle, sprue, runner into the cavity.

•The compressed gas takes path of the least resistance, flowing towards the melt front where the pressure is lowest to product light weight rigid parts.

•Advantages are No sink marks, reduced material cost, reduced cooling time, less warpage, rigidity for large structural parts etc.,

•Disadvantages are Air trapping, gas penetration into thin section, Short shot etc.,



SPECIALIZED INJECTION MOLDING

•**GAS-ASSISTED INJECTION MOLDING**:

1- cycle begins

1-2 Resin Injection stage

2- Resin Injection ends

2-3 Delay time

3 Gas injection begins

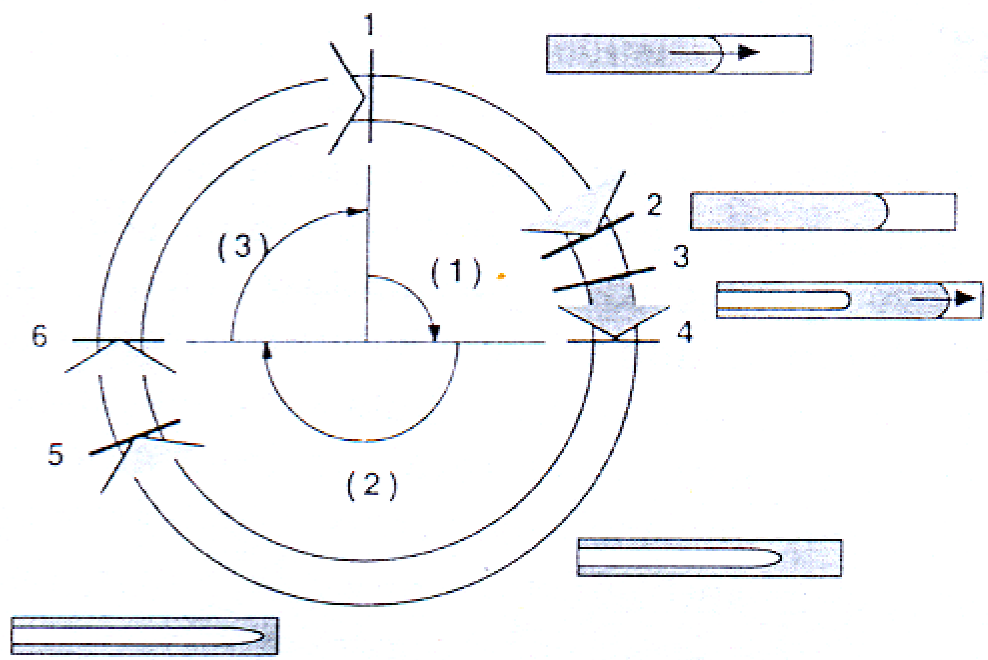
3-4 Gas injection during filling

4 Cavity filling completed

4-5 Gas pressure hold up

5 Gas pressure released

6 Mould opens



SPECIALIZED INJECTION MOLDING

•**GAS-ASSISTED INJECTION MOLDING**:

•**PRESSURE DISTRIBUTION:**