Mould : Mould can be define as a device from which plastic component are produced with the help of moulding machine .

Plastics can be classified according to manufacturing methods into two main groups:

1. Those which harden when heated as a result of chemical change (thermosetting or duroplastic materials).
2. Those which soften when heated , and solidify on cooling (thermoplastic materials).

The manufacturing process and the mould to be used are strongly influenced by these characteristics.

In the first case the material which hardens during chemical change is fed into the mould in a cool state or, in order to hasten the condensation process, slightly preheated. In this case the material is (in principle) cool, and the mould heated.

In the second case the material has to be heated. The softened material is injected in the cool cavity, where the material is shaped to the form of the cavity and hardened. In this case the material is heated and the mould cool.

Technology		Material to be used		Range of application
Name of method	Characteristic	Type	Form	
Compression moulding	Cool or slightly preheated material is formed in a hot mould with high specific pressure.	Thermosetting	Powder or preheated pellet	Oldest manufacturing method. Used for production of shaped pieces.
Transfer moulding	Cool or slightly preheated material forced with high specific pressure from a loading chamber into the mould cavity of a hot mould.	Thermosetting	Powder	For production of shaped pieces. Its productivity exceeds that of compression moulding with less tolerance.
Injection moulding	Heat-softened material injected with high specific pressure into the cooled mould cavity, where it settles.	Thermoplastic	Powder, grain	For production of shaped pieces.
Extrusion	Heat-softened material forced with great pressure through the orifice forming the profile of the work piece.	Thermosetting, thermoplastic	Powder, grain	For production of shaped rods, pipes, strips or thicker fibres.
Blowing	Heat-soften material formed with low pressure air in a cool (cooled) mould	Thermoplastic	Sheet , film	For production of hollow pieces.

Top clamping Plate:

Holds the stationary part of the mould to the stationary platen of the injection machine.

Location Ring:

Fits into a counter bore in the top clamping plate and is used to locate the mould on the platen of the press so the nozzle and sprue bushing are aligned.

Cavity Retainer Plate:

Part of the stationary section of the mould into which the leader or guide pins are mounted. Also used to hold core, cavity blocks, and sprue bushings.

Core Retainer Plate:

Top plate of the movable section of the mould. Forms the parting line of the mould with cavity retainer plate. Used to hold the leader pin bushing as well as core and cavity blocks.

Core back Plate:

Mounted behind the core retainer plate to keep this plate from bending under the high pressure used in injection molding:

Bottom Clamping Plate:

Holds the moving portion of the mould to the movable platen of the injection machine.

Spacers:

Mounted on the bottom clamping plate under the support plate to form a space which allows the ejection bar to move when the piece parts are ejected.

Ejector Retainer Plate:

Counter bored for the heads o0f ejector pins, ejector return pins, and spur puller pin.

Ejector back Plate:

Bolted together with the ejector retainer plate to form a unit. Acts as a back up plate for the pins in the ejector retainer plate.

Feed Buttons:

Pressed into the bottom clamping plate, they are lands for the ejector plate.

Pillars:

Round bars placed between the support plate and bottom clamping plate . The same height as the parallels. Bolted to the bottom clamping plate, they are used as additional support for the core retainer plate.

Sprue Bushing:

Butted up against the nozzle of the injection machine. Has a conical-shaped hole through which the material is forced into the mould runner.

Sprue puller Pin:

Pin located directly under the opening of the spur. Used to pull the moulded spur out of the bushing after shot has been made.

Push back pins:

Located in the ejector retainer plate. Force the ejector plate and ejector retainer plate, and therefore the ejector pins, to the bottom position as the mould closes.

Leader Pins /Guide Pillars:

Hardened and ground steel pins pressed into one of the plated. Align the two halves of the mould base.

Guide bushes:

Hardened and ground steel bushings which are pressed into one of the plates. Serve as bearing surfaces for the leader pins. Some injection mould bases are manufactured with the parallels welded to the bottom clamp plate. The unit thus formed is called the Ejector housing.

Cavity:

The female portion of a mould which gives to the moulding its external form.

Core:

The male portion of a mould which forms the internal shape of the moulding.

Parting Surface:

That part of the mould plate, adjust to the impression, which butt together to form a seal and prevent loss of plastic material from the impression.

Classification of Mould :

- Compression Mould :
  1. Hand Type
  2. Positive Type
     - Landed Positive
     - Direct Positive
  3. Flash Type
     - Semi + ve Vertical Flash
     - Semi + ve Horizontal Flash
     - Open Flash

- Transfer Mould :
  1. Pot Type
  2. Plunger Type

- Injection Mould :
  1. Based on Function
     - Hand
     - Semi Automatic
     - Automatic
  2. According to Construction
     - 2 –plate
     - 3-Plate
  3. According to Runner system
     - Cold Runner
     - Hot Runner
     - Insulation Runner

- Blow Mould :
  1. Extraction
  2. Injection
  3. Stretch

- Thermoforming Mould :
  1. Vacum
  2. Pressure

- Extrusion Mould :
  1. Film extrusion
  2. Pipe extrusion

- Roto Mould :

Injection Mould : The injection mould is an assembly of parts containing within it an 'impression' into which plastic material is injected and cooled. It is the impression which gives the moulding its form. The impression may, therefore, be defined as that part of the mould which imparts shape to the moulding.

The impression is formed by two mould members :

(i) The Cavity , which is the female portion of the mould, gives the moulding its external form.

(ii) The core, which is the male portion of the mould, forms the internal shape of the moulding

Work cycle of Injection Mould :

Closing the Mould - The clamping force is applied to close the mould on the projected area. The clamping force is obtained directly from hydraulic or through a toggle mechanism.

Injection - The injection ram comes forward along the heat plasticized material, is injected under pressure by a plunjer through a nozzle into the closed mould cavity. Via sprue bush, runners and gates. The plunger provides necessary pressure to inject the plasticized material.

Opening the Mould - The clamping force is relieved and mould is opened, side core if any one with drawn.

Ejection - As mould opens further the fixed stopper is provided in the machine to actuate the ejector system of the mould and the moulded pieces are ejected.

Day light of Mould : The basic mould consists of two parts or halves, namely a fixed and a moving half, respectively. When these two parts are opened the moulding can be extracted. Such an assembly is sometimes referred to as a single day light mould because, when the mould is open, there is only one space or, as it is normally termed, day light, between the two mould halves

.

One more complex moulds more than one day light occurs when the mould is opened. Consider , for example a stripper plate mould. This type of mould consists of three parts, namely a fixed mould plate, a moving mould plate and a stripper plate. When the mould is fully open there are two day lights. The moulding and feed system are removed from the first day light, that is from between the stripper plate and the fixed mould plate.

Another type of mould where more than one day light occurs. This mould also consists of three main parts a feed plate, a floating cavity plate and a moving mould plate. When the mould is opened there are again 2 day lights. This design permits a particular feed technique known as 'underfeeding' and the double day light is necessary in this case to permit the feed system to be removed from the mould. The mouldings are removed from between the floating cavity plate and the moving half while the feed system is removed separately from between the feed plate and the floating cavity plate. If the stripper plate and under feed designs are combined, as is often necessary for multi impression cap moulds. These mould is called a triple day light mould.

Two plate mould – If the mould separates into two main pieces or halves, on either side of the mould split line, then the mould is referred to as a two-plate injection mould.

Three plate Mould – This type of mould has three main parts or plates, which on opening are separated one from the other to give two day lights. The mouldings drop from one day light and the feed system, in the case of a cold runner type , drops from the other. One of the big advantages of this type of mould is that pin-point gates may be used for multi-impression moulds. Such gates may or may not be central with respect to the component.

The three main parts or plates which make up this type of mould are :

1. The stationary or runner plate or the clamping plate – which is attached to the stationary or fixed plate
2. The centre or floating plate – which is linked to the two other main plates.
3. The moving plate or the front cavity plate – which is attached to the moving platen of the machine.

Injection Moulding Method : Injection moulding normally involves use of thermoplastic materials and temperatures from 120° C to 300° C. Injection pressure usually run from 8000 to 20,000 psi. A two plate injection mould is shown in Fig . This mould will produce multi number mouldings during each shot or cycle Each cavity is filled by means of a feed system and this feed system , in the cash shown , consists of a sprue, runners and gates.

The sprue is usually a tapered circular channel through which the material flows after leaving the nozzle of the injection moulding machine. In the case of multi –impression moulds the cavities are joined to the sprue by means of runners. These runners are channels which are cut in one or both mould haves and their function is to fill each mould cavity as efficiently as possible. In between the runner and the cavity there is usually a narrow restriction in the runner system and this restriction (where the material enters the cavity) is called gate. (As the gate size is much smaller than the runner , the moulding may be easily separated from the feed system at this point when desired.)

During moulding the plastics material is forced through the sprue , into the runner system and from there into the cavity (or cavities). The component is formed between the opposing core and cavity, and the surface finish on the component is dictated by the surface finish on these two mould parts.

The half of the mould part moves with the clamp is referred to as the moving-mould half and provision is usually made to ensure that the moulding is retained in this half of mould. This is because the moulding ejection system is associated with the moving-mould half for reasons of cost, convenience, etc. In the mould shown , the moulding is made to stay with the moving-mould half by using what is known as a 'sprue puller'. Because this sprue puller has a re-entrant shape it dislodges, or pulls, the sprue from the sprue bush when the mould is opened. At the end of ,or during , the mould opening stroke the ejector system is activated and this clears the mouldings, and their associated runner system , from the mould. The depression immediately opposite the sprue, which contains the sprue-puller pin , is also known as the 'cold slug well' and its function is to accept this first part of the shot, i.e. that part of the shot which is located at the tip of the nozzle and which could therefore be cold.

To ensure that production is consistent and uniform the mould temperature is controlled , e.g. by circulating water through cooling channels (in the case of thermoplastics) or by means of electrical resistance elements (in the case of thermosets).

Die Materials :

The following factors should be considered in selecting materials :

a) Wear resistance : The ability to resist abrasive wear.
b) Machinability and / or hobbability : The ease with which the material may be cut or removed in machining or displaced in cold hobbing.
c) Toughness or shock resistance : The ability of a steel to withstand impact or to resist concentrated stresses which might result in premature brittle failures.
d) Surface cleanliness : The ability to be polished to a high luster without imperfections.
e) Compression strength : The ability of the die material to withstand high pressure without mechanical failure.

Material Used for Mould

Name of the Parts	Material	Hardness
Top plate	M.S / ST.42	
Bottom plate	,,	
Ejector plate	,,	
Ejector back plate	,,	
Core / Cavity holding plate	,,	
Support plate, Back plate	,,	
Spacer, Locating ring	,,	
Feet button, Anchor	,,	
Guide Pillar / Bush	OHNS / EN31 / 17Mn 1 Cr 95	C.Hdn HRC 58-60
Ejector Rod / Bush	,,	,,
Dowel Tube	,,	
Sprue Bush	OHNS / T110 W2 Cr 1	HRC 52-55
Sprue Puller	,,	HRC 54-56
Ejector Pin / Sleeve	,,	HRC 54-56
Wearing plate	,,	
Core Insert	En 24 / T110 W2 Cr 1	HRC 54-56
Cavity Insert	En 24 / T110 W2 Cr 1	HRC 52-55

Mould Making :;- A competent mould maker must have a through knowledge of the principle of mould making as the design of the various parts of the mould depends on the technique adopted for its manufacture. The majority of mould are manufactured by the use of conventional machine tools found in most modern tool rooms. From the manufacturing view point we classify the mould into two parts. 1. The cavity and core and 2. The remainder of the mould. The latter part is commonly referred to as bolster work.

The work on the cavity and core is by far the most important as it is from these parts that the plastics moulding takes its form. The work on the cavity and core can further be classified depending upon whether the form is of a simple or a complex nature.

The bolster work is not as critical as the manufacture of the cavity and core forms but, nevertheless, accuracy in the manufacture of the various parts is necessary to ensure that the mould can be assembled by the fitter without an undue amount of bench work.

Bench Fitting : Irrespective of the machine tool or technique used to manufacture the various parts of the mould, the final responsibility for the finishing of the individual parts and for fitting them together lies with the bench fitter. The mould finishing and assembly procedure adopted by the bench fitter varies from tool room to tool room and quite often between individual tool makers working in the same tool room; it is therefore impossible to set down a standard pattern for the work.

The various stages in the bench fitting involved in the manufacture of a simple mould are –

Stage 1. Finishing the impression
Stage 2. Aligning cavity and core
Stage 3. Bedding down
Stage 5. Water cooling circuit
Stage 6. Fitting ejector system
Stage 7. Fitting sprue bush and register ring
Stage 8. Polishing , hardening and try-out

The Technique of Injection Moulding :
In injection moulding the raw material is plasticized outside the mould in a separate cylinder where it is heated. The plasticized material is then injected through a nozzle under pressure by means of a plunger or a screw. The material enters the mould through channel and occupies the cavity. The plastic material is then held under pressure in the cavity and allowed to cool by means of cooling water circulating in the mould. After the mould has cooled and solidified the mould is opened and the moulded part is removed.

Core and Cavity Plates : Every mould consists of two plates called core and cavity plate. Into one plate is sunk the cavity which shapes the outside form of the moulding and is therefore known as the cavity plate. Similarly the core which projects from the core plate forms the inside shape of the moulding. When the mould is closed , the two plates come together forming a space between the cavity and core which is the impression.
The moulding , as it cools , will shrink on to the core and remain with it, as the mould opens. This will occur irrespective of whether the core is in the fixed half or the moving half. How ever, this shrinkage on to the core means that some form of ejector system is almost certainly necessary. Motivation for this ejector system is easily provided if the core is in the moving half.

Methods of incorporating cavity and core : There are various methods by which the cavity and core can be incorporated in their respective halves of the mould. These represent two basic alternatives –
a) The Integer Method where the cavity and core can be machined from steel plates which become part of the structural build up of the mould. When the cavity or core is machined from a large plate or block of steel and used without bolstering as one of the mould plate, it is termed an “integer cavity plate” or “integer core plate”. This design is preferred for single impression moulds, because of the strength, smaller size and lower cost characteristics.
b) The Bolster Method where the cavity and core can be machined from small blocks of steel termed inserts and subsequently bolstered. For mould containing intricate impressions, and for multi-impression moulds, it is not satisfactory to attempt to machine the cavity and core plates from single blocks of a steel as with integer moulds. The machining sequences and operation would be altogether too complicated and costly. The inserted bolster assembly method is therefore used instead

Venting in Injection Moulds :
When plastic material enters an impression air is displaced. Normally the air can escape between the two mould plates. However, should the plates have a very fine lapped finish the air may be trapped with in the impressions so that moulding defect, such as discoloration, sinks, incompletes filling etc. develop.
It is good practice to provide vents in the mould to allow air to escape freely. It is not usually to predetermine where the vent will be required. Normally vents are machined into mould plate once the mould has been tried out.
The vent is normally a shallow slot, not more than 0.05 mm deep by 3mm wide, machined in the land.
Positions where a vent is likely to be required.

1. At the point further most from the gate on symmetrical mouldings
2. At the point where flow paths are likely to meet
3. At the bottom of projections or blind recesses in cavities.

Shrinkage : Shrinkage is an inherent property of the polymer. When plastic is heated it expands. Upon cooling to the same temperature it will contract to the same volume. Shrinkage on cooling causes either sink mark on the outside or void on the inside. Sink mark are the result of insufficient material at a given spot. Shrinkage is defined as the difference between corresponding linear dimensions of the mould and the moulded article, both measurements being made at room temperature. Normally the shrinkage is expressed in %.
Ways to vary shrinkage : It is possible to vary the shrinkage a little by the condition of running after following rules :

1. Increased packing pressure reduce the shrinkage
2. Extend packing pressure time reduce the shrinkage
3. lower mould temperature reduce the shrinkage
4. Higher material temperature reduce the shrinkage
5. Slower sprue speed reduce the shrinkage.

Parting Surface :

The parting surface of a mould are those portions of both mould plate, adjacent to the impression, which butt together to form a seal and prevent the loss of plastic material from the impression.
We can classify the parting surface as being either flat or non-flat.
In general , the flat parting surface is the simplest to manufacture and maintain. It can be surface – ground, and is easily bedded down.

To bed down a pair of mould plates is the process of marrying the two mould surface together. This is accomplished by bluing one surface, momentarily bringing the two plates together and subsequently removing any spots which will be apparent on the non-blued surface. The plates are said to be bedded down when an even film of blue is transferred from one plate to the other.

Flat Parting Surface : The nature of the parting surface depends entirely on the shape of the component. The parting surface must be chosen so that moulding can be removed from the mould easily. The parting line must occur along the line round the position of maximum dimension when viewed in the draw direction.
Non-Flat Parting Surface : Many mouldings are required which have a parting line which lies on a non-planar or curved surface. In these cases the mould's parting surface must either be stepped, profiled or angled.

Balancing of mould surface : When the parting surface is not flat, there is the question of unbalanced forces to consider in certain instances. The plastic material when under pressure within the impression, will exert a force which will tend to open the mould in the lateral direction. If this happens some flashing may occur on the angled face. The movement between the two mould halves will be resisted by the guide pillers, but even so because of the large forces involved , it is desirable to balance the mould by reversing the step so that the parting surface continues across the mould as a mirror image of the section which includes the impression. It is often convenient to specify an even number of impressions when considering this type of mould, as impressions positioned on opposite sides of the mould's centre-line serve to balance the mould. When balancing is not practicable, due to size, then very sturdy guide piller must be incorporated.

EJECTION ;
All thermoplastic materials contract as they solidify, which means that the moulding will shrink on to the core which forms it. This shrinkage makes the moulding difficult to remove. It is normal practice, therefore, to provide some means by which the moulded part can be positively ejected from the core. Facilities are provided on the injection machine for automatic actuation of an ejector system, and this is situated behind the moving platen. Because of this, the mould's ejector system will be most effectively operated if placed in the moving half of the mould, i.e. the half attached to the moving platen.

Ejector Grid : The ejector grid is that part of the mould which supports the mould plate and provides a space into which the ejector plate assembly can be fitted and operated. The grid normally consists of a back plate on to which is mounted a number of conveniently shaped ' support blocks'. There are three alternative designs – i) The in-line ejector grid ii) The frame-type ejector grid iii) The circular support block grid

Ejector Plate Assembly : The ejector plate assembly is that part of the mould to which the ejector element is attached. The assembly is contained in a pocket, formed by the ejector grid, directly behind the mould plate. The assembly consists of an ejector plate, a retaining plate and an ejector rod. One end of this latter member is threaded and it is screwed into the ejector plate. In this particular design the ejector rod functions not only as an actuating member but also as a method of guiding the assembly. The parallel portion of the ejector rod passes through an ejector rod bush fitted in the back plate of the mould.

Ejector plate assembly return system : The ejector plate assembly is moved forward relative to the remainder of the moving half. Now technique is to be provided to return the ejector plate assembly to its rear position in preparation for the next shot, when the mould closes.
Two system in common use are i) the push-back return system and ii) the spring return system.

Stop pins : With a large ejector plate or large ejector bar system, it is often preferable to incorporate stop pins on the underside of the ejector plate. This design drastically reduces the effective seating area. In doing so , it diminishes the possibility of the ejector elements remaining slightly proud of their correct position due to foreign matter being trapped behind the ejector plate.

Ejection Techniques : The basic ejection techniques are – i) pin ejection ii) sleeve ejection
iii) bar ejection iv) blade ejection v) air ejection vi) stripper plate ejection.

Sprue pullers : When the mould opens it is essential that the sprue is pulled positively from the sprue bush. With single impression moulds the sprue feeds directly into the base of the component and the sprue is pulled at the same time as the moulding is pulled from the cavity.
For multi impression moulds using a basic feed system the sprue would probably be left in the sprue bush each time the mould was opened. This would necessitate a manual operation to remove the unwanted sprue. To avoid this undesirable feature, an arrangement for pulling the sprue should always be incorporated in the design. The common sprue pulling methods utilize an undercut pin or an undercut recess situated directly opposite the sprue entry. The plastic material which flows into the undercut, upon solidifying, provides sufficient adhesion to pull the sprue as the mould is opened.

Pin Ejection : This is the most common type of ejection as , in general, it is the simplest to incorporate in a mould. With this particular technique the moulding is ejected by the application of a force by a circular steel rod, called an ejector pin. The ejector pin is headed to facilitate its attachment to the ejector plate assembly. In operation, the ejector plate assembly, to which the ejector pin is attached, is moved forward relative to the mould plate. Thus the ejector pin pushes the moulding from the cavity. The working diameter of the ejector pin must be good slide fit in its mating hole in the mould plate. If it is not, then plastic material will creep through the clearance and a mass of material will progressively build up behind the mould plate. The direction of movement of this ejector pin is , therefore , controlled by this hole. The plain diameter ejector pin may be used in one of the two ways. It may be used as a moulding face pin, in which case the whole of the top surface of the pin is in contact with the moulding. The alternative is to use the ejector pin as a parting surface (butting face) pin. In this case only a part of the top surface of the pin is in contact with the moulding ; the rest abuts on to the fixed mould plate when the mould is closed. The location of the ejector pin elements, and the number used, is dependant on the component's size and shape. The ejector pins should be located, therefore, so that the moulding is pushed off evenly from the core.

Valve Ejection : The valve ejector element is basically a large-diameter ejector pin. Valve-type ejection is used, normally, for the ejection of relatively large components in situations where it is impracticable to use standard parting surface pins. The valve-type ejector element is design to apply the ejector force on to the inside surface of the moulding. However, with valve ejection , because the element has a large effective ejection area, the risk of the moulding being distorted is minimized. The ejector element incorporates a valve-type head which is seated in a nest machined into the core.

Air Ejection : With this ejection method the ejector force is provided by compressed air, which is introduced directly on to the moulding face via a small air ejctor valve. For this method to operate efficiently the adhesion between the moulding wall and the core must be broken locally, to permit the compressed air to be introduced. This is achieved by causing the valve ejector to move forward slightly by air pressure. The introduction of the compressed air can be under either manual or machine control.

A few advantages of air ejection

i) No ejector grid or ejector assembly is required. This reduces the cost of the mould.
ii) Air ejection can be fitted in either mould half.
iii) The ejection can be operated at any time during the opening stroke of the machine.
iv) The air-valve ejector element acts as a vacuum breaker between the moulding and the core. Connection to an external compressed air supply is not necessary.

Limitations :

i) The method is only effective on certain types of component.
ii) A compressed air supply must be readily available.
iii) Air is an expensive service and if incorrectly used this system can wasteful.

Sleeve Ejection : With this method the moulding is ejected by means of a hollow ejector pin, termed a sleeve. It is used in one of three circumstances :

1) For the ejection of certain types of circular moulding.
2) For the ejection of circular bosses on a moulding of any shape.
3) To provide positive ejection around a local core pin forming a round hole in moulding.

The sleeve , which is a sliding fit in the cavity insert and on the core pin, is fitted at its rear end to the ejector assembly. The core pin extends completely through the sleeve, and is attached to the back plate. When the ejector assembly is actuated , the sleeve is moved relative to the core (and to the cavity) and the moulding is ejected. Ejection by means of a sleeve is a particularly efficient method because the ejection force is applied to a relatively large surface area.

Stripper Plate Ejection : This ejection technique is used primarily for the ejection of circular box- type mouldings. The stripper plate is mounted between the cavity plate and core plate. The aperture in this stripper plate is a sliding fit on the core. When the mould starts to open the stripper plate moves back with the core plate. Once the moulding is clear of the cavity, the movement of the stripper plate is arrested, while the core plate continues the rearward movement. The core is thereby withdrawn through the stripper plate and the moulding ejected. The mould is then closed in preparation for the next cycle.

Stripper Bar Ejection : This method is an extension of the parting surface ejector pin principle, in which the ejector element is caused to push against the bottom edge of the moulding. However, a far greater effective ejection area is obtainable with the stripper bar method and because of this characteristic, this method of ejection is particularly suitable for thin-wall mouldings. The stripper bar is fitted into a complementary shaped recess in the mould plate. A small angle of 10 degree is incorporated all around the periphery of the stripper bar to minimize wear. The stripper bar is coupled to the ejector plate by a a tie-rod. The stripper bar should be manufactured from a nickel-chromium steel and suitable heat treated.

INJECTION MOULDING MACHINES

The process of injection moulding essentially comprises of plasticizing the raw material in a cylinder by the application of heat and then injecting it under pressure through a nozzle by means of a ram into a closed mould, where it is allowed to cool and then opening the mould and removing the moulded component.

The injection moulding machine essentially consists of the units namely, the mould clamping unit and the injection unit. The mould clamping unit performs the functions of clamping the mould, locking the mould and opening it. The injection unit plasticizes the material and injects it into the mould.

There are two methods employed to inject the plastic material into the mould, one is by means of a plunger and another by means of a screw which transports the materials during which time it is plasticized and then is injected. Though there are the plunger type machines in operation, the modern trend is to use a screw which is to be lending itself for better controls.

Injection moulding machines are made in a wide range of shot capacity ranges from a few grames to about 30 kgs and the mould locking capacity from 15 tons to 5000 tons.

Types of Injection Moulding Machines : Injection moulding is being done in machine shop in different types of machines.
The main types are –
1. Hand injection moulding machine.
2. Piston type injection moulding machine.
3. Screw type injection moulding machine.

Hand injection moulding machine : The hand injection moulding machines which are available in our country have a capacity of 2 ozs. They are generally being used without thermo regulators. The temperature controls provided along with the machine under the category thermostat which are only indirect controls. The major limitations and advantages of hand injection moulding machines are as follows –

1. Variation of shrinkage.
2. Appearance of sink marks.
3. Variation in cooling time.
4. Variation in mould temperature.

With all these limitations, the injection moulding by hand is still a consumer of large amount of thermoplastics in our country because the capital involved is less. As it is labour oriented this method can be successfully followed when the labour is cheap in a country like ours.

Piston type injection moulding machine : The piston type injection moulding machine is nothing but a cold piston moving inside a hot cylinder which contains plastic materials. In order to facilitate dispersion of heat and get a homogeneous melt the cylinder is provided with a torpedo. Actually when the piston moves forward the molten material diverges at one end of the torpedo before getting shot through the nozzle.
Limitation and advantages of piston type injection moulding machine –
Limitations :

1. Inability to control injection speed.
2. Contamination while change over.
3. Lack of homogeneity of the melt.

Advantages :

1. Fast cycle
2. Easier operation
3. Less cost
4. Possibility of packing

Screw type injection moulding machine : The screw type injection moulding machine consists of a screw rotating in a hot barrel capable of delivering continuous melt of plastic. By operating this screw we can get an intermittent flow of plastic through nozzle. Because of the shearing action and material flowing in different patterns there is a homogeneity in the melt. From this we can observe that the melted pieces will have better properties when processed in a screw type injection moulding machine.

Especially while moulding engineering items and thick walled items, and items of very large dimensions it is advantageous to use screw type injection moulding machines to have a precise control over all the required properties in an injection moulded product.

The limitation with the above machine are

1. Low cost
2. Longer cycle time

The difference between screw machine and plunger machine in feeding, melting and injecting the material.

Feeding – The plunger machine has a feeding machines which controls the flow of pellets into the heating cylinder. In the screw machine, the pallet fall by the gravity into the flights of the screw.

Melting – The plunger machine melts the plastic by heat conduction from the cylinder walls. A spreader is used to thin out the plastic, so more plastic is put in contact with the wall of the heating cylinder causing faster melting. In the screw machine, melting is caused by both heat from the cylinder walls and frictional heat from shearing and squeezing of the plastic along the flight of the screw.

Injection – In the plunger machine pressure is exerted by the injection plunger to force the melt into the mould. In screw machine, the screw stop rotating and is then used as the ram to force the plastic through the nozzle into the mould.

Comparison of the relative advantages of each system (Bolster & Integer type)

Cost : The cost of mould material – The integer method requires the whole mould plate to be made of expensive mould steel, whereas the insert-bolster method needs only that part which forms the impression to be made of mould steel.
The machining and fitting of a single-impression integer type of mould is less costly in time and in the number of operations as compared with the insert-bolster combination.
Number of impression : The difficulty in machining and aligning the cavities and cores in an integer type mould increases with the number of impressions the mould contains. Therefore for multi-impression moulds it is usually preferable to use the insert-bolster system.
Multi-impression mould alignment : Misalignment etc if detected can usually be rectified in an insert-bolster assembly. It is far more difficult in an integer mould.
Mould size : The overall size of an integer mould will be smaller than a corresponding insert-bolster mould. However the integer mould requires that quite heavy steel blocks are handled during the manufacturing stage.
Heat treatment : During the heat treatment the possibility exist that the steel may distort. The smaller the block of steel the less likely is the distortion to occur. Thus, from the hardening stand point , the insert method is preferred to the integer mould.
Replacement of damaged parts : With the insert-bolster system it is possible to repair a damaged impression without much interruption to production.
Cooling system : This is usually far simpler to design for an integer cavity or core plate because the designer can place his cooling system close to the cavity wall without the sealing complications that arises when attempting to cool cavity and core inserts.
Conclusion : Unquestionably for single impression moulds the integer design is to be preferred irrespective of whether the component form is a simple or a complex one. The resulting mould will be stronger, smaller. Less costly, and generally incorporate a less elaborate cooling system than the insert-bolster design.
For multi impression mould the choice is not so clear cut. In the majority of cases the insert-bolster method of constructions is use, the ease of manufacture, mould alignment , and resulting lower mould costs being the overriding factors affecting the choice.

Type of bolster : -

i) Solid bolster – This is suitable for use with both rectangular and circular inserts.
ii) Strip type bolster – Suitable only for rectangular insert.
iii) Frame type bolster – Although this can be used for both types of inserts, it is particularly suitable for circular inserts
iv) Chase bolster – This type is used in conjunction with ‘splits’
v) Bolster plate – This is used in particular circumstances with certain types of both rectangular and circular inserts.

Runner System : The runner is a channel machined into the mould plate to connect the sprue with the entrance (gate) to the impression. In the basic two- plate mould the runner is positioned on the parting surface while on more complex designs the runner may be positioned below the parting surface. The wall of the runner channel must be smooth to prevent any restriction to flow.
Following points must be considered while designing runner system –

1. The shape of cross section of the runner
2. The size of the runner
3. The runner layout

The Shape of Cross Section : The cross sectional shapes of the runner used in a mould are usually fully round, half round, Hexagonal, trapezoidal, modified trapezoidal.
The criterion of efficient runner design is that the runner should provide maximum cross sectional area from the standpoint o f pressure transfer and a minimum contact on the periphery from the standpoint of heat transfer. The ratio of cross sectional area to periphery will, therefore, give a direct indication of the efficiency of the runner design; the higher the value the greater the efficiency.
The points concerning the choice of runner cross-sectional shape , we can say that for simple two-plate moulds which have a flat parting surface the fully round runner or hexagonal runner is to be preferred , the increased mould cost being relatively small. For moulds which have complex parting surfaces , where it would be difficult to match accurately the semicircular channels of the round runner or for multi plate moulds, the trapezoidal or modified trapezoidal section should be used.

Cooling of Moulds : One fundamental principle of injection moulding is that hot material enters the mould , where it cools rapidly to a temperature at which it solidifies sufficiently to retain the shape of the impression. While the melt flows more freely in a hot mould , a greater cooling period is required before the solidified moulding can be ejected. Alternatively , while the melt solidifies quickly in a cold mould it may not reach the extremities of the impression. A compromise between the two extremes must therefore be accepted to obtain the optimum moulding cycle. To maintain the required temperature differential between the mould and plastic material , water (or other fluid) is circulated through holes or channels within the mould. These holes or channel are termed flow-ways or water-ways and the complete system of flow ways is termed the circuit.

Adaptors : The majority of moulds are drilled to provide a flow-path through which the coolant can be circulated. These drillings are connected to the supply and return lines via adopters. The adopter is a standard mould pipe fitting which can be obtained in number of a lternative design and sizes.
A typical basic design of adopter – One en d of the adopter is threaded to correspond to that of the flow-way tapped hole, the other end incorporates either : i) External serrations for use with PVC house, utilizing hose clips for attachment purposes ; ii) Theards, for use with a corrugated metal hose system with in-built threaded couplings. This system offers advantages in that it is more leak free in operation, and it will withstand higher pressures.

The sizes in common use are as follows :
BSP : 1/8, 1/4, 3/8 , 1/2, 3/4

O-Ring : An O-ring (O-seal) is a synthetic rubber ring which is incorporated in a suitable recess in a mould for the purpose of preventing leakage of the coolant fluid. For this function to be achieved effectively, the O-ring must be suitably compressed by a specific amount in order to achieve the required leak-free joint. Two alternative cross sectional shapes are available, namely circular and rectangular.

Standard Mould System :
A mould system may be defined as an assemblage of mould parts, the plates of which conform to an accepted structural shape and size. The mould system may be purchased either in kit form or as an assembled mould unit. Naturally the mould plates in the above mould kit or mould unit do not contain impression form, as this aspect of mould manufacture must be left to the specialist mould maker.
The two-part mould is adopted as the standard mould system by most manufacturers because this particular mould construction is the most widely used design in industrial practice. The system comprises two mould plates (a cavity plate and a core plate) plus an ejector system, guide pillars, guide bushes etc.

Advantages and Limitations of Standard Mould Unit :

Advantages :

1) Less steel needs to be carried in stock, therefore investment is reduced.
2) Buying and stock control are simplified.
3) The cost of the mould unit is known, therefore estimating is easier.
4) Waiting time for steel blanks, etc., is avoided.
5) Shaping, planning and drilling of steel plates and blocks I s avoided.
6) Turning, grinding and fitting of guide pillars and bushes is similarly avoided.
7) The ejector plate is pre-positioned and located.
8) Machine time is saved as a result of 5 and 6 above.
9) Labour time is saved as a result of 5 – 7 above.
10) Work on the impression can usually begin immediately.
11) The individual mould unit components are standard: if damage occurs during manufacture or in production, a part can be quickly replaced.
12) In small moulding companies, one mould unit can be used for several similar jobs. Only the impression inserts need to be changed.
13) The overall time the mould is in the tool room is reduced.
14) Mould delivery time is reduced.
15) Highly paid mould-makers may be employed on important impression work rather than on relatively minor 'bolster work'.

Limitations :

1) The number of sizes available is limited.
2) Maximum size available is relatively small.
3) Maximum depth of mould plate is also relatively small.
4) The ejector stroke may be larger than is actually required.
5) The positioning of coolant flow-way holes is often made difficult by pre positioning of guide pillars, guide bushes, screws and dowels.
6) The support blocks are positioned relatively far apart. Extra support blocks may have to be fitted if deflection of the mould plate is to be avoided.
7) The back plate cannot be unscrewed independently of the support block and backing plate. To expose the ejector assembly, the mould half must be disassembled.

Manufacturers :

1) DMS -Diemould Service Company Ltd, commonly known as DMS. This is a British company.
2) DME – Detroit Mold Engineering Co (USA)
3) Hasco is a West German Co.
4) Uddform (in Sweden, West Germany, and japan)
5) Desoutter is a British Co.

Splits :

A moulding which has a recess or projection is termed as undercut moulding. Any recess or projection on the outside surface of the component which prevents its removal from the cavity is termed as external undercut. There are two forms of undercut to be considered. i) The undercut may be local, in that the recess or projection occurs in one position only. The clip on a pen cap is an example of this. ii) The undercut may be a continuous recess or a projection on the periphery of the component. The water connector has a number of such undercuts.

In either case, it is necessary to split the cavity insert into parts and open these, generally at right angles to the line of draw, to relieve the undercut before the moulding is removed.

Since the cavity is in two pieces, a joint line will be visible on the finished product. Now this joint line, on an undercut component, is comparable to the parting line on an in line of draw component and the same careful consideration must be exercised in deciding its position before attempting to design the mould. However it is desirable to keep the splits movement to a minimum the joint line for regular rectangular components should be positioned on the longitudinal centre-line. The split can be incorporated in the mould design in several ways. Generally the splits are retained on the mould plate and actuated automatically. There are two basic designs: sliding splits and angled splits. In both designs there are moving parts and it is necessary to arrange for i) guiding the splits in the desired direction ii) actuating the splits, and iii) securely locking the splits in position prior to the material being injected into the mould.

Methods of operation : There are various methods which can be used to actuate the splits in relation to the mould plate.

The most frequently used designs are based on various types of cam. e.g finger cam, dog-leg cam, and cam track methods of actuation.

The basic operation with cam actuation : As the mould is opened, the cams attached to the fixed half cause the splits to slide across the moving mould plate. Conversely, when the mould halves are brought together, the splits are progressively closed. The cams generally lose contact with the splits as the mould opens and should either split be moved out of position prior to the mould closing serious damage will occur. To obviate this danger some safety features are incorporated in the design.

Another method of actuating the splits is by the use of compression springs, but as these can only be used to open the splits, the locking heels on the mould plate are utilized to fulfil the closing function.

Most machine manufacturers incorporate facilities in the hydraulic circuit for the operation of additional actuators if required. Thus providing an actuator can be accommodated in the design, this method of splits operation should not be overlooked. The actuators can be mounted on opposite sides of the mould and the rams coupled directly to the splits. The main advantage of this system is that large splits movement are practicable.

Finger Cam Actuation : In this system, harden , circular steel puns, termed finger cams, mounted at an angle in the fixed mould plate. The splits, mounted in guides on the moving mould plate, have corresponding angled circular holes to accommodate these finger cams.
As the mould opens, the finger cam forces the split to move outwards, sliding on the mould plate. Once contact with the finger cam is lost, the split's movement ceases immediately. Continued movement of the moving half causes the ejector system to operate and the moulding to be ejected. On closing the reverse action occurs. The finger cam re-enters the hole in the split and forces the split to move inwards. The final closing nip on the splits is achieved by the locking heels and not by the finger cams.

The distance traversed by each split across the face of the mould plate is determined by the length and angle of the finger cam. the movement can be computed by the formula
$M = (L \sin Ø) – (c / \cos Ø)$
As the required movement is known from the amount of component undercut, the following rearranged formula to determine the finger cam length is of greater use, apart from checking purposes.
$L = (M / \sin Ø) + (2c / \sin 2 Ø)$
Where M = splits movement
Ø = angle of finger cam
L = working length of finger cam
C = clearance.
The designer must aim to keep the splits' movement down to a minimum, at the same time ensuring that the moulded part can be easily and quickly removed from the mould. The clearance c serves a dual purpose. It ensures that the force which is applied to the split during the injection phase is not transferred to the relatively weak cam. it permits the mould to open a predetermined amount before the splits are actuated. 10 degree is a suitable angle for Ø , But it is permissible to increase this angle up to a maximum of 25 degree. One or two finger cams are used to operate each split.

Dog-leg Cam Actuation : This method of actuation is used where a greater splits delay is required than can not be achieved by the finger cam method. The dog-leg cam which is of a greater rectangular section, is mounted in the fixed mould plate. Each split incorporates a rectangular hole, the operating face of which has a corresponding angle to that of the cam.
The sequence of operation : As the mould is closed and the splits are locked together by the locking heels of the fixed mould plate, the splits do not immediately start to open when the mould halves are parted because of the straight portion of the dog-leg cams. The moulding, which is encased within the splits, will thus be pulled from the stationary core. Further movement of the moving mould half causes actuation of the splits by the dog-leg cams, thereby releasing the moulding. The reverse action occurs when the mould is closed.
The relevant formula for calculating the opening movement, the of cam and the delay period are

$M = La \tan Ø - c$
$L_a = (M+c) / \tan Ø$
$D = (L_a – e) + (c / \tan Ø)$
Where M = movement of each split
L_a = angled length of cam
L_s = straight length of cam
Ø = cam angle, c = clearance D = delay
e = length of straight portion of the hole.

The cam is held back is on to the mould plate by a socket-headed screw.

Spring Actuation Method :

A simple method of actuating angled splits is by the use of springs. The forward movement of the splits is limited and because of this feature the design is used for components which have comparatively shallow undercuts.

A bolt is fitted to the locking angle of the split. The bolt extends through a plate securely attached to the mould plate. A spring is mounted between the plate and the head of the bolt. Here, again the strength of the spring must be such that it normally holds the splits open and it does not put undue strain on the cams during the closing stroke.

A stop pin is used to control the maximum movement of the split.

Loose Threaded Cores :

In cases when a large component incorporates a local internally threaded hole, the loose threaded core technique should be considered. The threaded hole in the moulding is being formed by the loose threaded core. When the mould is opened, the moulding is ejected by an ejector pin system. The loose core is then unscrewed and replaced for the next moulding cycle. Two sets of loose cores should be used during production. At the end of the first moulding cycle, the second set of cores can be inserted into the mould and the next cycle commenced.

Mouldings (product) Defects and Causes :

Flash - Inadequate tightness of mould parts , Clamping pressure too low , Injection too fast , Polymer too hot.

Burning - Gate too small, Inadequate venting , Claming pressure too high , Injection too fast , Polymer too hot.

Jetting - Improper gate location , Gate too small , Polymer too cold , Injection too fast

Bubbles (voids)- Inadequate venting , Thin to thick transition , Mould too cold, Pressure too low , Polymer too hot , Lack of holding pressure

Sinks - gate too small , Inadequate venting , Mould too hot , Pressure too low , Polymer too hot

Silver streaks - Gate too small , Inadequate venting , Insufficient drying of resin , Polymer too cold , pressure to low

Delaminations - Mould too cold , Incompatible polymer

Weld lines - Mould too cod , Polymer too cold Injection speed to slow

Gate splay - Improper gate location , Gate too small , Mould too cold , Polymer too cold , Injection too fast

Warping - Cooling too short , Polymer is too hot , lack of cooling

Errors in mould and product design with possible consequences for process and / or moulded part.

Faults	Possible Problems
Wrong location of gate	Cold weld lines, flow lines, jetting, air entrapment, venting problems, warping, stress concentrations, voids and /or sink marks
Gates and / or runners too narrow	Short shots, plastics overheated, premature freezing of runners, sink marks and / or other marks
Runners too large	Longer moulding cycle, waste of plastics, and pressure losses.
Unbalanced cavity layout in multiple-cavity moulds	Unbalanced pressure build up in mould, mould distortion, dimensional variation between products (shrinkage control poor), poor mould release, flash and stresses
Non uniform mould cooling	Longer moulding cycle, high after-shrinkage, stress (warping), poor mould release, irregular surface finish, and distortion of part during ejection
Poor or no venting	Need for higher injection pressure, burned plastic (brown streaks), poor mould release, short shots, and flow lines
Poor or no air injection	Poor mould release for large parts, part distortion, and higher ejection force
Poor ejection system or bad location of ejectors	Poor mould release, distortion or damage in moulding, and upsets in moulding cycle.
Sprue insufficiently tapered	Poor mould release, higher injection pressure, and mould wear
Sprue too long	Poor mould release, pressure losses, longer moulding cycle, and premature freezing of sprue
No round edge at end of sprue	Notch sensitivity (cracks, bubbles, etc) and stress concentration
Bad alignment and locking of cores and other mold components	Distortion of components, air entrapment, dimensional variations, uneven stresses, and poor mould release
Mould movement due to insufficient mold support	Part flashes, dimensional variations, poor mould release, and pressure losses.
Radius of sprue bushing too small	Plastic leakage, poor mould release, and pressure losses
Mould and injection cylinder out of alignment	Poor mould release, plastic leakage, cylinder pushed back and pressure losses
Draft of moulded part too small	Poor mould release, distortion of moulded part and dimensional variations
Sharp transition in part wall thickness and sharp corners	Part unevenly stressed, dimensional variations, air entrapment, notch sensitivity, and mould wear

Compression Mould :
Compression moulds make use of a mold cavity for receiving the compound when the mold is open, and a force or plunger for compressing the compound as the mold closes. These moulds are generally used for the thermosetting and cold-moulded materials only. The mould may be either hot or cold. They are used hot with the thermosetting phenolics, ureas, etc. And cold with the cold-moulding compounds. Compression moulds are seldom used for the thermoplastic materials because of the long period required for heating the material to the plastic stage and the necessity for chilling the mould immediately thereafter in order to harden the piece.

The fundamentals of mouldings :
PLATEN : The flat surface of the moulding press on to which the two halves of the mould are mounted. Generally positions horizontally on compression presses and vertically on injection machines.
DIE or CAVITY : female section of the mould which generally the external shape to the part to be moulded.
PUNCH or PLUNGER : The male portion of the mould. In compression moulding the plunger pushes or forces the material into any opening in the cavity. Gives the internal shape to the part being moulded. In injection moulding, the plunger is often referred to as the “Core”
CHASE : Steel plates which are used to hold cavities and plunger
MOULD : a) To shape or form an article b) The entire mechanism-cavity plungers, chase etc.needed to mould plastic articles.
MOULDING : The article or part to be moulded.
STOPPER : A horizontal surface to limit stroke of press.
PARTING LINE : Line at which the two half of the mould, plunger and cavity meet.
FLASH : A thin section or fin that forms at the parting line.
TABLET or PREFORM : Moulding material that has been pressed into some desired shape for efficient loading of the mould. The tablet can have a form which is near to the finish moulded piece. e.g. round, ablong, square.
Advantages :
Eliminate larger raea of filling room, Correct quantity of powder is maintained, For multicavity mould, loaqding becomes easy, More effective in pre-heating because they are more homogeneous

Flash Mould :
The cavity is filled with material and the excess is squeezed out over the lands which are about 3 to 5 mm wide. External stoppers are provided so the plunger does not crush the top of the cavity, when the mould is completely closed. Clearance between 0.1 to 0.3 mm is provided so that excess material can escape and the cavity is not damaged.
In this type the excape avenue is an open one with no restrictions other than the narrowing of the avenue as the land faces are approaching each other. It is often used for flat articles such as switch plates, deep bowls, breakers, but it is not the best type to use because of the waste of powder involved.

Types of Moulds :

1. Positive mould with horizontal flash
2. Positive mould with vertical flash
3. Semi-positive mould with horizontal flash
4. Semi-positive mould with vertical flash

1 & 2 are very old methods used in olden days. When it was guessed that material in the mould become fluid before it was cured and would escape if not prevented. It was used in early days when designs were the results of guessed work. These moulds will work moderately well with "phenolics" but are not at all very successful on urea formaldehyde or melamine formaldehyde materials. The important thing is that a positive mould is seldom as good even on phenolocs as a semi-positive mould.

The dis-advantages of this design are that the variation in the amount of material loaded will result in variation of thickness so that the weight of the charge must be accurately controlled. Further, the gases liberated during the moulding process are trapped inside as they have no free means of escape and may show as blisters or blemishes on the moulded surface.

3 & 4 are the semi-positive editions of 1 and 2 positive mould with vertical and horizontal flash and are much better mould from the application point. The satisfactory working of these semi-positive mould is applied by having clearance between the plunger and cavity 0.025 mm for 25 mm dia. Further , to give an ever increasing path to the escaping flash the mould is given a taper between 2 to 3 on each side. This gives a chance for the flash to flow on and also entrapped gasses along with it, thereby producing a clean blemish-free moulded component.

Basic Principle in Compression Moulding is that the plastic material becomes fluid under the influence of heat and pressure and the material is forced into any opening in the mould to obtain the desired shape.

Compression moulds are made of steel. The moulding section of the moulds are harden and highly polished. The two halves of the mould are mounted between the platens of a hydraulic press. The material is placed in the cavity of the heated mould in the form of a powder or perform. As the two halves of the mould are closed by the press, the plunger causes the material to flow through the mould . As the material is compressed into shape , the heat and pressure causes a chemical reactions in the material which is set or cured into the shape of the required piece part.

Heating of Compression Mould : Thermo-set materials which are used in compression moulding are cured by heat and pressure. Heating of compression moulds is an important phase in the moulding operation. Heating of moulds using thermo-set material serves the purposes :

i) Heat must softer the material sufficiently to allow it to flow under influence of the press pressure, into any opening in the mould to the desired shape.

ii) Enough heat must be applied to bring about the chemical change co polymerize the material into its hard, infusible finish state.

Factors that must be considering in Compression moulding are Temperature , Pressure and Cure time.

Temperature : Temperature for moulding TS materials vary from 130º C degree to 180º C. Temperature for moulding the various materials can be determined by experimentation or by getting the information from the manufacturer of some particular material. Temperature which are too high causes blisters and burn spots on the finished article. Temperature which are too low do not allow the material to flow properly and results incompletely filled cavities and in sufficiently cured piece parts. Mould temperature must be maintained for best results. Moulding temperature not only vary with the material used but with the geometry of the moulded article, the type of mould, and whether loose powder or pre-heated pre-forms are used.

Method of heating : The two methods most commonly used in heating (press platens) and moulds using thermo-set materials are steam and electrically. Steam is favored by many moulders because of its economy and heating qualities. Electric heating of moulds is replacing steam in many instances because of its cleanliness and lower maintenance costs.

Pressure : Pressure needed to mould a particular article depends on the flow characteristics of the material, the cavity depth, and the projected area of the piece part. Generally it is recommended that minimum mulding pressure of 200 kg / cm2 of projected area to be used. However to compensate for any variables that may be encountered.

Cure Time : Cure time varies with the material used, the size and shape of the moulded article, and the method of moulding. In compression and transfer moulding, cure time is the elapsed when the movement of the press stops until the pressure on the moulded part is released. For smaller and thin wall pieces, cure time may be only a minute or two. On larger pieces, and pieces with thick sections, the cure time may be as high as 15 minutes. Curing time always has a connection with mould temperature. High temperature chort curing time Low temperature low curing time.

Vents : During the process of compression moulding , gases are formed as the chemical reaction takes place in the material. Some provisions must be made to get rid of the gases. Gas pockets can cause in-complete shotes or blistered piece parts. One method of getting rid of the gases is to allow the mould to breathe, that is the mould is closed and then opened again for about 3 mm to get rid of the gases and then closed again. Other methods are to make small flats ground on the periphery of the plunger that telescopes into the cavity. These flats will have 0.05 to 0.25 deep and about 3 mm wide, ground into top position of the cavity well. This type of construction is used on large cavities.

Drafts : Draft is provided for the removal of parts from the mould. It is possible when absolutely necessary to produce some surface without draft. A minimum of ½ degree taper per side is generally satisfactory, although 1 degree per side is most desirable for production job side wall taper of large units must permit the flow of material from the bottom of the mould, upto the parting line when compression moulds are used.

Breathing : After full pressure is first reached, the pressure has to be released for a few seconds to allow the entrapped gases to escape. This is known as ‘Breathing’. The duration and the extent of breathing has to cooled out or found out to suit the particular mould and moulding condition.

Common faults that are met with in compression moulding, their causes and remedies.

Condition of Piece	Probable Cause	Possible remedy
1. Pieces blistered	Moulding cycle too short Gasses trappe in mould Mould too hot or cold Insufficient pressure Piece designed with thick non-uniform walls	Increase time of cycle Provide grooves to allow escapement of gas Pre-heat material Reduce temperature or increase temp. Increase pressure Tablet the charge Pre-heat matl Reduce moulding temperature
2.Moulded piece is flexible on discharge	Piece is insufficiently cured Material has absorbed water by exposure Too much lubricant used on mould	Increase curing time If temp. is low increase it Pre-heat material Do not use lubricant
3.Moulded piece sticks	Temp. too high and resin fails to come to surface and cover it Mould has un descent by scratches , dents Mould is poorly polished Distortion caused by knockout pins due to flexibility of piece	Reduce temperature. Repair mould Polish Mould Increase curing or use material with more rigidity or discharge.
4. Surface of piece dull	Mould too hot or cold Mould improperly polished Mould stained by previous material Poor grade of steel used for mould	Correct temperature Polish mould Clean and polish mould Polish and chromium plate the cavity
5.Surface of piece orange peeled	Mould closed too fast under too high pressure Material too soft A number of perform used for charge Material contains	Close mould more slowly Use harder material Use one perform. Pre-heat material
6. Pieces warped	Mould heated non uniformly Material too soft Insufficient cure Mould too hot causing case hardening on pieces Pieces of irregular wall thickness and shape resulting in and shrinkage	Correct heating method Use harder material Increase cycle Decrease temperature a) Use lower shrinkage material b) Redesign piece to uniform section c) Pre heat the metal.
7. Piece crack at once or afterward in storage	Wall around insert too thin Pieces strained on ejection Where piece is cooled too much in mould	Increase wall and use lower shrinkage material Eject evenly Reduce cooling cycle
8. Surface has hard small spots on pieces similar to small blisters, commonly known as pimples	Material contain foreign matter Material has hard particles in it or is too course.	Protect against contaminator Use finer material

<u>Transfer Mould</u> : Transfer moulding is another method of moulding thermoset material. The material is not placed in the cavity as in compression moulds, but is heated in a loading chamber and enters the closed mould in a fluid state by the action of a plunger or hydraulic cylinder. The fluid material is directed to the cavity or cavities by a sprue, runners and gates. Pressure is kept on the material until the part is cured. There are a number of advantages of transfer moulding thermosetting material –

i) has a shorter moulding cycle than compression moulding.
ii) It is capable of moulding closer tolerances
iii) It moulds easy, thin walled section
iv) There is less strain on small core pins
v) Material flows better around intricate protrusions and inserts.

Defination
Sprue : A round taper opening from the loading chamber which feeds the material into the cavity or into the runners.
Runner : A groove or channel varying in size and depth, through which the fluid material flows to the vicinity of the cavity.
Gate : A narrow opening in the cavity which carries the material from the runner into the cavity.

There are two basic types of transfer mould construction :

1. Pot transfer
2. Plunger transfer

Pot Transfer : The pot transfer is used in the conventional compression moulding press. The pre-form is loaded into the pot or loading chamber. The material is heated by the hot mould and combined with the pressure of the pot plunger, the material becomes fluid and is forced into the cavity to form the piece part. Pressure is kept on the material until the piece parts are cured. Excess material forms a cull at the bottom of the pot and also forms a dove tail shape in the plunger called the cull pick up.
After the parts have been cured, the mould opens. As the plunger comes out of the pot, the sprue is broken at the small diameter of the taper, the cull pick up on the plunger carries the cull out of the pot and the sprue out of the sprue bushing. Upward of the press continues to open the downward movement of the plunger plate is halted , while the rest of the mould below the parting line continues to move. The mould is constructed so that the moulded pieces remain in the cavities. Continued movement of the press activates the knockout bar and the moulded part are ejected from the cavities.
To protect the edges of the plunger, a soft headed mallet is used to drive off the cull pick up and cull from the plunger. The piece parts are removed from the mould and the runner broken off at the gates.
All excess material is removed from the cavities, pot and plunger. The mould is closed until the parting line is mated and the mould is again in loading position.

Plunger Transfer Mould : The plunger type transfer mould in the closed position with the preforms in the target area and the plunger on its down ward stroke. The combination of the heat of the mould and the pressure of the plunger on the preforms causes the material to become fluid and to flow through the runners and gates into the cavities. The plunger transfer differs from the pot transfer in that the plunger is part of the moulding press and not a part of the mould itself. By the use of this plunger, the sprue is eliminated and very thin cull of small area is formed above the target area, thus reducing loss of material. The top clamp plate is fastened to the stationary platen of the press.

It is recommended that a clamping pressure 700 to 800 kgs per sq. cm be used to keep the mould together at the parting line. Pressure on the transfer plunger is generally about 550 kgs per sq. cm. All transfer moulds must be vented to allow air to escape from the cavities.

Designing of Pot and Plunger : Both the pot and plunger are made from a good grade of wear resistant tool steel which is heat treated and ground. Pots and plungers are made square, rectangular or round in shape. The shape is determined by the shape of the piece part, no. of cavities and available space in the mould base. Round pots and plungers are preferred because less machining difficulties are encountered. A clearance of 0.025 to 0.075 mm/ side is provided between the pot and plunger

The area of the pot should be 20% to 30% greater than the area of all the cavities and runners. The dimensions of the pot, if it is round or square, can be calculated once the area is known.

To determine the volume of the pot, the total volume of all the piece part, the runner and the sprue, plus a small amount for a 0.375 to 0.75 mm thick cull, is approximately calculated. At least twice this volume is to be used for the pot volume. Knowing the area and the volume one can easily arrive at the depth of the pot by dividing the volume by the area. The additional volume in the pot is provided to compensate for the bulk factor of the performs used and to allow the plunger to enter the pot a short distance before exerting pressure on the material. High bulk factor materials are generally not used in transfer moulding. The bulk factor of the performs used in transfer molding is approximately 1 to 2 figs. (illustrate some of the construction details of the pot and plunger.

For sufficient strength, horizontal distance “Y” should be equal to the depth of the pot “Y” , - A1.5 to 3 mm radius is provided at the top edge of the pot. A1.5 mm radius is machined at the bottom of the pot to facilitate the flow of the material and to simplify the machining of the corner. A2.5 to 3 mm radius is machined at the bottom of the plunger. The difference in the radii on the plunger and the bottom of the pot results in a clearance so that the plunger will not wedge in the pot but will land on the flat surface of the pot. In assembly there is a small clearance between the plunger and the transfer pot.

In practice it was very difficult to maintain the clearance for long time. Fitting the plunger to the cold pot size – the plunger staying relatively cold during operation but the pot having to heated – the clearance widened between them allowing material outflow.
Adjusting the plunger to the chamber when expanded by heat prevented its entrance in the cold state was difficult. So it frequently occurred that the plunger was forced into the pot strongly on clamping the mould and there was seizing the cracking and other damage occurred.
This problem is solved by adjusting the plunger to the cold pot with running fit, but permitting material flow around the plunger so as to form
A collar when solidified according to the hot pot size, thus preventing material flowout, during molding. A sealing groove approximately 2.5 mm wide and 0.8 mm deep is cut into the perimeter or periphery of the plunger.
During the operation of the mould this groove fills with the molding material and acts as a natural sea, allowing very little material to escape past the bearing surface of the plunger. Flats or grooves are ground on the bearing surface of the plunger for venting purposes. A clearance of 0.75mm per side is machined above the bearing surface of the plunger. This clearance keeps the bearing surface narrow to prevent galling, and allows flash and excess material to escape. The sprue and the interior or the pot is polished so the material can flow easily. The sprue has a taper of 2 to 3 per side. The large diameter of the sprue varies in size from 9 to 12 mm with a 1.5 to 3 mm radius at the entrance of the sprue. The small diameter (at the runner or piece part) varies from 3 to 6 mm depending on the size of the piece part. Wedge-shaped slots called cull pickup are machined in to the plunger. The thick or heavy section of the cull pickup is located directly above the sprue as

shown in fig (T2.3). The length of the cull pickup varies from the width is generally 2 to 3 times the diameter of the sprue.

CALCULATION OF POT DIMENTSION IN TRANSFER MOULD

The clamping pressure provide by the chamber is an important consideration,

If the total cavity area is greater than the total pot area, the hydraulic pressure exerted by the plastic compound would tend to open the mould at the parting line.

So insure perfect mould locking, the area (Ap) should 25% to 30% greater than the combined area of the molding surface and the area of all runners and sprues.

The dimension of pot,

If it is round or square can be calculated , once the area is known.

Total area of pot Ap

= total projected area of cavities, runners and sprues + 25% to 30% of total projected area.

Volume of pot Vp

= total volumes of all the piece parts, runners and sprues + approximate volume of a small amount of .5 to 1 mm thick cull multiplied by bulk factor of the compound.

Depth of pot = Vp / Ap

<u>Common faults that are met with in Transfer moulding, their causes and remedies</u>.

Fault	Cause	Remedy
Short moulding	Insufficient moulding powder	Increase charge to pot
Blisters or soft moulding	Under cured	Increase cure time
Excessive flash	Transfer pressure too high	Reduce pressure or increase locking force
Gas marks	Trapped gas	Pre-heat material prior to charging pot
Burn marks	Air trapped in mould	Arrange for proper venting
Pre-cure	Pot or mould temp. too hot Pre-heating temp. too high	Reduce temperature Reduce cycle time Reduce temp. of pre-heating
Ripples or orange peel	Un-even flow in mould	Use free flowing material, apply more pressure

Hot-Runner unit Moulds :
This is the name given to a mould which contains a heated runner manifold block within its structure. The block, suitably insulated from the rest of the mould, is maintained at a closely controlled elevated temperature to keep the runner permanently as a melt. The polymer material can thereby be directed to the mould extremities without loss of heat and without the pressure loss associated with temperature variations. The hot runner unit is mounted adjacent to the cavity plate and accommodated in a suitably designed grid. The polymer material enters via a centrally positioned sprue bush, passes through the flow-way and leaves the unit via a secondary nozzle in line with the impression. When the mould is opened the moulding is pulled from the cavity, and the sprue is broken at the small diameter end. The remainder of the feed system remains heated within the unit, ready for the next shot.

<u>Advantages of the Hot Runner Mould</u>:

1.) There is no feed system for the operator to remove from the mould.
2.) On manually controlled injection machines the mould open time is reduced.
3) The cost of separating, storing and regrinding scrap feed systems is thereby saved.
4) As no scrap material is produced it cannot be contaminated.
5) The moulding is automatically degated on ‘direct feed’ designs.
6) All of the impressions fill at practically the same time.
7) Thin wall-section mouldings fill relatively easily because the melt is at a higher temperature close to the impression.
8) The diameter of the flow-way within the hot- runner unit can be larger than a corresponding runner machined in the parting surface of a two-part mould. There is, therefore, less pressure loss throughout the system.
9) Mouldings are produced with less inherent strain because lower pressure are required.
10) As there are no runners (in the conventional sense) to feel and cool, the time cycle is marginally shorter with the hot-runner unit design. This advantage is most pronounced with respect to multi-impression moulds.
11) Improved moulded-part quality is obtained, particularly where large gates can be incorporated.
12) The hot-runner mould is not restricted to use on a particular injection machine.
13) The location of the impression is not confined to the central region of the mould plate, as is the case when a small manifold nozzle design is adopted.
14) On conventional runner-type moulds the weight of the feed system must be taken into account when determining the number of impressions which can be moulded on a specific size of injection machine. This is unnecessary with the hot-runner unit design, and therefore more impressions can be incorporated.

<u>Limitations of the Hot-Runner Mould</u>:

1) The mould setting time is generally greater than for a corresponding two-part mould.
2) An extended period, waiting for the hot-runner unit to heat, is required before production can commence.
3) The initial ‘debugging’ of a new hot-runner unit mould is useally more extensive than with a standard mould.
4) The cost is higher than that of a standard mould and in some cases than that of an underfeed mould.
5) The area of the moulding adjacent to the gate may be blemished with surface heat marks.
6) Polymer melt leaking from the hot-runner unit can create problems.
7) Polymer material at the gate may solidify and interrupt production.
8) Certain materials (nylon) have a tendency to drool (dribble) from the gate into the impression when the mould is open. This causes blemishes on the subsequent mouldings.
9) Some materials (PP) have a tendency to ‘string’ when the moulding is extracted from the cavity. This has adverse effects similar to the above.
10) There is a tendency for certain types of heating element to fail during production.
11) Replacement of the heating elements is sometimes difficult and can be time-consuming.

12) With certain materials (PVC) there are degradation problems.
13) Changing the colour and the type of grade of material can also be a problem.
14) Heat expansion of the hot-runner unit can create difficulties, if not allowed for.

Blow Moulding Principle : The blow moulding process is used to make hollow articles, especially bottles, barrels, and other liquid containers. A tubular perform , called a parison, is either injection moulded or extruded and cut to length the extrusion method is the more useful.
The hot parison is placed in a split hollow and blown up to conform to the contour of the mould by air pressure and it touches the cold walls of the mould , it cools down and become rigid. The mould is then opened and the part is removed.
Since only air pressure must be resisted by the mould, aluminium is the only mould material. Aluminium also offers the advantage of rapid cooling of the article because of its high thermal conductivity. The blow moulding process has four basic operations –

3. Production of the parison
4. Positioning of the mould half to entrap the parison.
5. Forming the neck of the container.
6. Injection of air and cooling of the mould.

Thermoforming of plastic film and sheets : Thermoforming is the process of heating a plastic material in sheet form to its particular processing temperature and forming the hot and flexible material against the contours of a mold by mechanical means (e.g. differential in air pressure created by pulling a vacuum or using the pressure of compressed air). When held to the shape of the mould and allowed to cool, the plastic retains the shape and detail of the mould. Because softening by heat and curing by the removal of heat are involved , the technique is applicable only to thermoplastic material and not to thermosets.
Example of thermoformed product are plastic or foam dinnerware , cups , trays, egg cartons, refrigerator liners, computer housings, interior and exterior automotive parts, blisters for packaging etc.
Advantages of thermoforming over most other methods of processing plastic include lower tooling and machinery costs, high output rates, the ability to use pre-decorated plastic sheet, and good quality physical properties in finished parts.
Its disadvantages include the need to begin with sheet or film rather than less costly basic resins, trimming material used to clamp sheet for forming, and the problem of trim scrap reclamation.

Rotational Moulding : In rotational moulding, the product is formed from liquid or powdered thermoplastic resin inside a closed mold or cavity while the mould is rotating biaxially in a heating chamber. Rotational moulding (also popularly known as rotomoulding) is best suited for large, hollow products requiring stress-free strength, complicated curves, a good finish, a variety of colors, a comparatively short production run, and uniform wall thickness. It has been used for products such as fuel tanks, furniture, tilt trucks, industrial containers, storage tanks, portable outhouses, modular bathrooms, telephone booths, boat hulls, garbage cans, light globes, ice buckets, appliance housings, and toys.

SHOT CAPACITY

The plunger type machine is often rated in terms of maximum SHOT WEIGHT (in gm) with Polystyrene. The rating in terms of another material can be determined as follows:

Shot capacity with material B

$$= (\text{Shot capacity with material } \mathbf{A}) \times \left(\frac{\text{Density of } \mathbf{B}}{\text{Density of } \mathbf{A}}\right) \times \left(\frac{\text{Bulk factor of } \mathbf{A}}{\text{Bulk factor of } \mathbf{B}}\right)$$

Where, **A** = Polystyrene
B = Plastic to be used

The screw type machine is normally rated in terms of SWEPT VOLUME (in cc) of the injection cylinder.

Shot capacity = **Swept volume (cc) X ρ (gm/cc) X c**

Where, ρ = Density of plastic material at normal temp. In gm/cc
c = 0.85 for crystalline Material.
= 0.93 for Amorphous Material

Material	Bulk factor	Material	Bulk factor
ABS, ACETAL, ACRYLIC	1.8 - 2.0	PP	1.9 - 1.96
SAN	1.9 - 2.5	POLYTHENE	1.8-2.3(LD) 1.7-1.9(HD)
NYLON	2.0 - 2.1	POLYSTYRENE	1.9 – 1.96
POLYCARBONATE	1.75	PVC	2.3 (Rigid) 2.2 (flexible)

Plasticizing capacity

Plasticizing rate of B (m/hr) = **Plasticizing rate of A (gm/hr) X $\frac{QA}{QB}$**

Where, A - Polystyrene
B - Plastic to be used.

QA -Thermal Capacity of A in cal/gm
=Specific Heat in (cal/gm) X moulding temp. Of A
QB - As same as A for B

Plasticizing rate (gm/hr)
= wt of moulding (gm) X (number of mouldings / hr)

For maximum efficiency machine should not operate above 80% of its rated shot weight or plasticizing capacity

Cycle time (tc) when press is limited by plasticizing capacity is given by

$$tc = \frac{m \times 3600}{P \times K}$$

Where, m = weight of shot (g)
P = plasticizing capacity with polymer to be moulded and
K = 1000 (metric unit).

MATERIAL	SP. HEAT	MOULDING TEMP.	MOULDING TEMP.	SP. HEAT	MATERIAL
ABS	0.35	225	250	0.38	NYLON-6
ACRYLIC	0.035	225	300	0.30	PC
ACETAL	0.35	225	240	0.55	PE
PP	0.46	250	180	0.24	PVC
PS	0.32	200	200	0.35	CAB
SAN	0.33	220	195	0.36	CA

PLATEN SIZE

This includes

- Distance between tie – bars
- Maximum Day lights
- Size of mould plates
- And other details of platen from machine manual

CLAMPING FORCE

The clamping force required to keep the mould closed during injection must exceed the force given by the product of the opening pressure in the cavity and the total projected area of all impressions and runners.

For plunger type Machines

Clamping force (tons)

= **Projected area of mouldings** Including runners in **cm^2 X 1/2 to 1/3 of injection pressure in kg / cm^2**

For Screw type machines

2/3 to 1/2 of injection pressure should be taken.

INJECTION PRESSURE

The injection pressure may be obtained from manual or may be calculated as follows:

Injection pressure (kg / cm^2)

$$= \text{Hydraulic line gauge pressure (kg / cm}^2\text{)} \times \frac{(\text{Inj. Cylinder Dia in cm})^2}{(\text{Heating cylinder ram dia in cm})^2}$$

$(Di^2)/(Dp^2)$ = 20 on many plunger type machines

DETERMINATION OF NUMBER OF CAVITIES

By Shot capacity

$$N_s = \frac{0.85\,W}{m}$$

Based on 85%of rated shot capacity

By Plasticizing capacity

$$N_p = \frac{0.85 \times P \times T_c}{3600m}$$

Based on 85%of rated plasticizing capacity

By clamping capacity

$$N_c = \frac{C}{P_c \times A_m}$$

.

Where,

N_s, N_p & N_c = Number of cavities based on shot, plasticizing and clamping capacity respectively.

W = Rated shot capacity for polymer in gm

m = Moulding wt per cavity in gm. (Volume* Density)

P = Rated plasticizing capacity for polymer (gm/hr)

T_c = Over all cycle time (sec)

C = Rated clamping capacity in tons

P_c = Clamping pressure in Tons/cm²of projected area (0.630T/cm²)

A_m = Projected area of moulding including runner (cm²)

Procedural steps during assembling of mould

Check : Part list – Assembly drawing , Part drawing , Parts , Quantity , Heat treatment parts ,Screw , Special devices (Arc medium screw , Gears , Latch , Hydraulic devices)

Instruments : Vernier caliper , Micrometer , Surface table , Parallel block , hammer (Copper, Leather) , Scraper , Prussian blue , Oil stone , Crane

- Study the drawing
- Inspection visually , Instruments
- Chamfering , Cleaning screw and other holes
- Cavity & Core polish by file , polishing elements
- Checking cooling circuit , Blocking with water connectors , o-ring , M-seal , Buffle blade , Pipe fitting and adopter , Nozzle
- Leak checking with water connector
- Pillar bush alignment separately
- Inserts match separately
- Collar mating in the plates with pins , round bar , inserts etc.
- Ejector system checking separately
- Insert fitting and Keying
- - Bolster work
- Incase of split mould – Guide way fitting and check Cam movement with pin , dog leg , spring , heel block ec. Ejector pin length , sleeve , ejector ring height etc . Calculation and grind
- Bedding down without cam and inserts
- Bedding down with cam , pin and inserts
- Fitting of sprue , Register ring , knockout rod
- Checking ejector system
- Air vent , gate cutting
- Try out