International Journal of Emerging Technology and Innovative Engineering
Volume 1, Issue 9, September 2015 (ISSN: 2394 – 6598)

DESIGN AND DEVELOPMENT OF FAMILY INJECTION MOULD FOR INNOVATIVE PLASTIC IN AND INNOVATIVE PLASTIC OUT

Y.Swathi[1], Dr.P. Mallesham[2] ,M. Kamal[3]

1. M.Tech Sri Indu College of Engineering and Technology
2. Principal Sri Indu College of Engineering and Technology
3. M.E University college of Engineering, Osmania University

ABSTRACT

Design and Development of Family Injection Mould for Innovative Plastic in and Innovative Plastic out is presented. This paper is fully concentrates how to design the injection mould by using the least economic processing and tooling method and also foresee the possible problem for a product design in Injection moulding. able to optimize the mould design process. The material 60 Glass Fiber (GF) NYLON6 is used to manufacture the component using injection moulding process. The Numbers of factors affect the quality of the component in an injection moulding are injection pressure, injection time and cooling time.

I. Introduction

Injection moulding is a manufacturing process for producing parts by injecting material into a mould. Injection moulding is one of the chief processes for producing plastic items. It is a fast process that can deliver a large numbers of identical items - from high precision engineering components to small disposable consumer goods. The use of this process has enabled the extension of boundaries of design in plastics, and also helped in the substitution of conventional materials, thereby reducing the weight of the end product and providing immense design freedom.

Injection moulding is used for making parts with all kinds of shape from simple to high complex, microscopic to very bulk ones. All this is done in single step process which is fast, consistent, safe, reliable and cost effective. The material formed would not need further finishing works. It is the process that is capable of production moulded parts of relatively intricate configuration with good dimensional accuracy. Injection moulding is a method of forming or moulding objects from powdered or granular type of thermoplastic material. Thermosetting powdered materials are also being injection moulded.

Injection moulding machines consists of a material hopper, an injection ram or screw type plunger, and a heating unit. They are also known as presses, they hold the moulds in which the components are shaped. Presses are rated by tonnage, which expresses the amount of clamping force can exert. This force keeps the mould close during the injection process. Tonnage can vary from less than 5 tons to 6000 tons, with the higher figures used in comparatively few manufacturing operations. The total clamp force is needed is determined by projected area of the part being moulded. This projected area is multiplied by a clamp force of from 2 to 8 tons of each square inch of projected areas.

As a rule of thumb, 4 or 5 tons/in^2 can be used for most products. If the plastic material is very stiff, it will require more injection pressure to fill the mould, thus more clamp tonnage to hold the mould closed. The required force can also be determined by the material used and size of the part, larger parts require higher clamping force.

1.1 Innovative plastic:

Plastics have inspired and continue to drive innovations that help to society's greatest challenges from life changing technologies to more living sustainable living and many of the important things in between. Plastic materials like nylon, silicon, polyster and spandex. Modern plastics fuel designers imaginations and consistently allow for the creations of products.

II. Problem statement:

The main aim is to "Design and development of family injection mould for Innovative Plastic IN and Innovative Plastic OUT"

- This paper fully concentrates to how to design the injection mould
- To foresee the possible problem for a product design and therefore able to optimize the design in the mould design process.
- To achieve the minimum production cycle time.
- Trouble shooting.

2.1 Cycle of operation for injection moulding:

a) Plasticizing the resin

The cycle begins with the external plasticizing the resin an accumulating it in the forward section of the barrel. The heater bands maintain the melts temperature as the shot it builds up. The mould is closed. The cycle is typically timed so that there is minimal time between the mould and the next shot.

b) Injection molten resin

Once the shot is ready, valve is opened at the nozzle and the melt is quickly injected into the mould. This part of the process only takes a few seconds. As the melt enters the cavity, the displaced air is vented out through the hole for the ejection pins and along the parting line. Proper filling of the cavity is depended on the part design as well as good gate locating and design proper venting.

c) **Cooling the part**

This is the longest portion of the moulding cycle. Once the cavity is filled, the part is allowed to cool. If an accumulator is not used, the extruder continuous to push material into the mould and maintain proper amount of pressure until the material cools (or freezes). This is also controlled by timers and coolants.

d) **Mould rotating**

After completion of first stage injection process, mould cools for a moment and then mould open. After opening the core region rotates into 180^0 then mold closes. This angle will be perfect alignment.

e) **Ejecting the part**

Once the part has cooled (so that it will hold its shape out of the mould, and ejection pins won't deform the part), the mould is opened. The moving platen has moves backwards and the ejector pins strikes the rear plate (or 'ejector plate'), ejects the part.

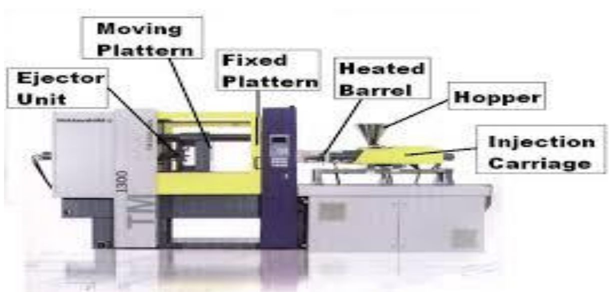


Fig.1 Injection moulding machine

2.2 Injection moulding machine components

1) Barrel
2) Heat control
3) Nozzle
4) Screw
5) Clamping system
6) Clamping force

2.3 Types mould for injection mouding

The mould can be designed in the matter of some factors, such as the dimension of the product and the size of the mould plate that is going to be used. It is usually ends on how many cavities will be possible to be made in one mould plate. Based on the cavity numbers, there are three types of moulds in the injection moulding.

1. Multi cavity mould

This type of mould is used when the product manufacture would like produce several identical products at each injection cycle. In this mould, there are several cavities in mould plate and each of them is inter connected by the runner system.

2. Family cavity mould

In this mould there are several cavities that are not identical. Each of the cavities represents a part of the finished product. So after each injection cycle, the parts needs to be assembled manually in order to obtain the finished product. In a small production unit this type of mould is normally avoided because the complexity it brings when designing the runner system to balance the flow melt. Also it requires bigger mould.

3. Single cavity mould

It is typically used for beginner until the experts in the field. This mould has only one cavity inside, quite simple in the design, and suitable for low production until high production plastic products.

Mould design

The mould consist of two primary components, the injection mould (A plate) and the ejector mould (B plate). Plastic resin enters the mould through a sprue in the injection mould the sprue bushing is to seal tightly against the nozzle of the injection barrel of the moulding machine and to allow molten plastic to flow from the barrel in to the mould, also known as cavity. The sprue bushing direct the molten plastic to the cavity images through channels that are machined into the faces of the A and B plates.

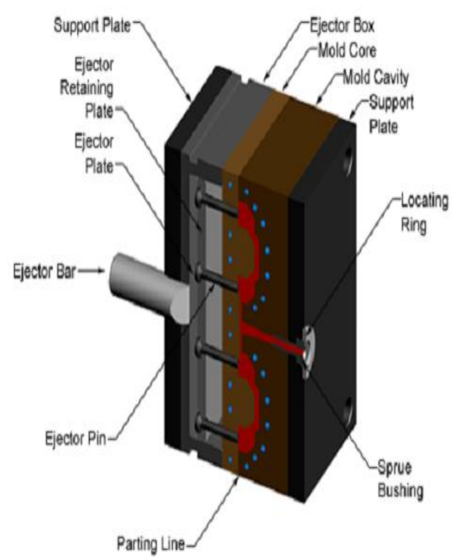


Fig.2 mould parts

That sprue bushing direct the molten plastic to the cavity images through channels that are machined into the faces of the A and B plates. These channels allows plastic to run along them, so they are refer to runners. The molten plastic flows through the runner and enters one are more specialized gates and into the cavity geometry to from the desire part.

The amount of resin required to fill the sprue, runner and cavities of a mould is a shot. Trapped air in the mould can escape through air vent that are ground in to the parting line the mould. If the trapped air is not allowed to escape, it*s compressed by the pressure of the incoming material and is squeezed into the corners of the cavity, where it prevents filling and causes other defects as well. The air can become so compressed that it ignites and burns the surrounding plastic material. To allow for removal of the moulded part from the mould, the mould must not overhang one another in the direction that the mould opens, unless parts of the mould are designed to move between such overhangs when the mould opens (utilizing components called lifters).

Sides of the part that appear parallel with the direction of draw (the axis of the cored portion) holes or insert is parallel to the up and down movement of the mould as it opens and closes are typically angled slightly with (draft) to ease release of the part from the mould. Insufficient draft required for mould release is primarily dependent of the depth of the cavity: the deeper the cavity, the more draft necessary. Shrinkage of must also be taken into account when determining the draft required. If the skin is too thin, then the moulded part will tend to shrink onto the cores that form them while cooling, and cooling those cores or part may wrap, twist, blister or crack when the cavity is pulled away.

The mould is usually designed so that the moulded part reliably remains on the ejector (B) side of the mould when it opens, and draws the runner and sprue of the (A) side along with the parts. The part falls freely when ejected from the (B) side. Tunnel gates, also known as submarine or mould gate, is located below the parting line. The moulded part is cut (by the mould) from the runner system on ejection from the mould. Ejector pins, also known as knockout pin, is a circular pin placed pin paced in either half of the mould (usually the ejector half), which pushes the finished moulded product, or runner system out of a mould.

The standard method of cooling is passing a coolant (usually water) through a series of drilled through the moulded parts and connected by holes to form a continuous pathway. The coolant absorbs heat from the mould (which has absorbed heat from the hot plastic) and keeps the mould at a proper temperature to solidify the plastic at the most efficient rate.

To ease maintenance and venting, cavities and core divided into pieces , called inserts, and sub assemblies, also called inserts, blocks, or chase blocks. By substituting interchangeable inserts, one mould makes several variations of the same part.

More complex parts are formed using more complex mould. These may have sections called slides that move into a cavity perpendicular to the draw direction, to form overhanging parts features. When the mould is opened, the slides are pulled away from the plastic part by using stationary “angle pins” on the stationary mould half. These pins enter a slot in the slits and cause the slides to move backward when the moving half of the mould opens. The part is then ejected and the mould closes. The closing action of the mould causes the slides to move forward along the angle pins.

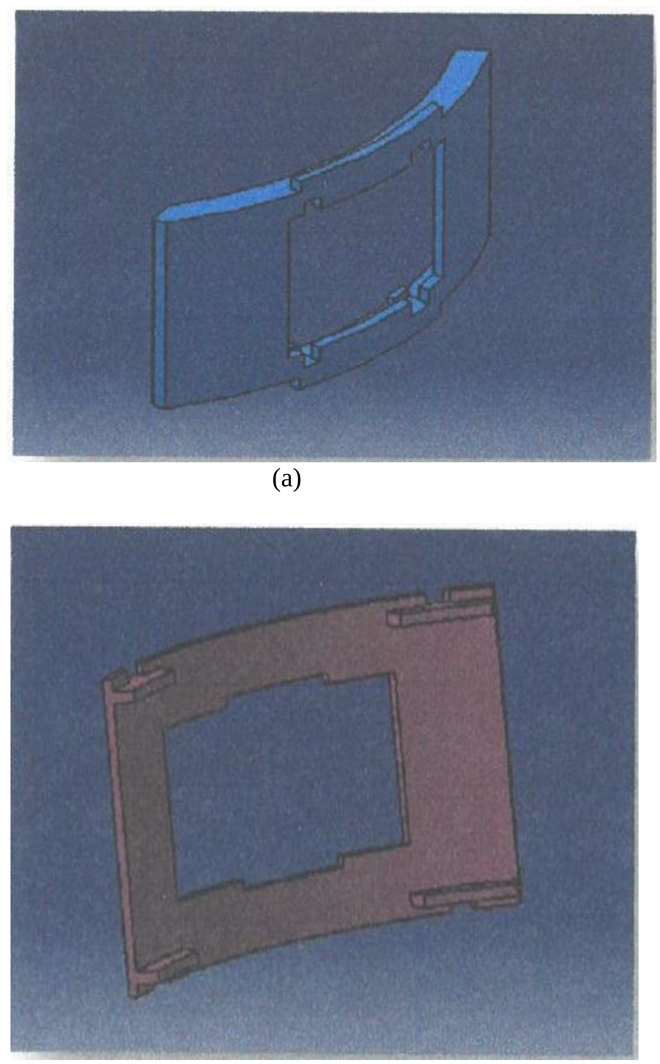
In this family mould there will be several cavities that are not identical. Each of the cavities represents a part the finished product. So after each injected cycle, the part needs to be assembled manually in order to obtain the finished product. In some cases multiple cavity tooling will mould a series of different parts in same tool. Some toolmakers call these moulds family moulds as all the parts are related.

2.4 Design of dies

Study of component

The following are the specification of component.

Component name	:	IP IN & IP OUT.
Tool number	:	IM: 382
Material	:	60% Glass Fiber NYLON 6.
Component weight	:	3.1 gm (App).
Shrinkage	:	0.2%
Gate	:	Edge gate



(a)

(b)

Fig.3 component drawing (a) Innovative Plastic IN and (b) Innovative Plastic OUT

Component material properties

Name	: 60% GF NYLON 6
Density	: 1.69 g/cc
Specific heat	: 0.38 cal/g/^{0}c
Melting temperature	: 220-262^{0}c
Shrinkage	: 0.2%
Injection pressure	: 88.5 mpa
Tensile strength	: 3.62(kg/cm^2) × 10^2
Compressive strength	: 4.2-7.04 (kg/cm^2) × 10^2
Coefficient of thermal expansion	: 9.0 mm/^{0}C × 10^{-5}
Thermal conductivity	: 0.0004 (cal/sec-cm-^{0}C)
Thermal diffusivity	: 9.45 × e^{-4} cm^2/sec^{0}C

The step by step procedure of mould design

1. Check the mould feasibility to design.
2. Identify critical and major dimensions.
3. Deciding parting line.
4. Add draft to the model considering tolerances for maximum material condition.
5. Add shrinkage to model.
6. Generate parting surface as per the deciding parting line.
7. Split core and cavity.
8. Internal splitting of core and cavity inserts considering manufacturing and assembly.
9. Check for draft analysis and clearances.
10. Feed system creation as per the standards (or) as per the customer requirements.
11. Create mould base.
12. Provide ejector positions in the component and inserts.
13. Create cooling holes.
14. Conduct concept review meeting.
15. Change the model or layout based on the review output.
16. Conduct assembly review meeting.
17. Prepare detail drawings of inserts and mould base elements.
18. Release drawings to manufacturing.
19. Collect document distribution list.
20. If any changes in the drawing then new drawings need to be released with new revision.

Design calculations

- IP IN component weight = 1.4 gm
- IP OUT component weight = 1.7 gm
- Component volume = Mass/Density
  = 3.1/1.69
  = 1.834 cc

Shot weight calculation:

- Component volume (v) = 1.834 cc
- Component density (d) = 1.69 gm/cc
- Shot weight = v × ρ
  = 1.834 × 1.69
  = 3.1 gm
- Total shot weight = shot weight + runner weight
  = 3.1 + 0.9= 4gm

2.5 Main elements of die

2.5.1 Inserts:

The following are the main inserts which forms cavity of component when assembled.

(a) Cavity insert:

It is the insert which has depression, or a set of matching depressions, into which material is injected which usually, forms the outer surface of the moulded part. This is rigidly fixed in the cavity insert plate.

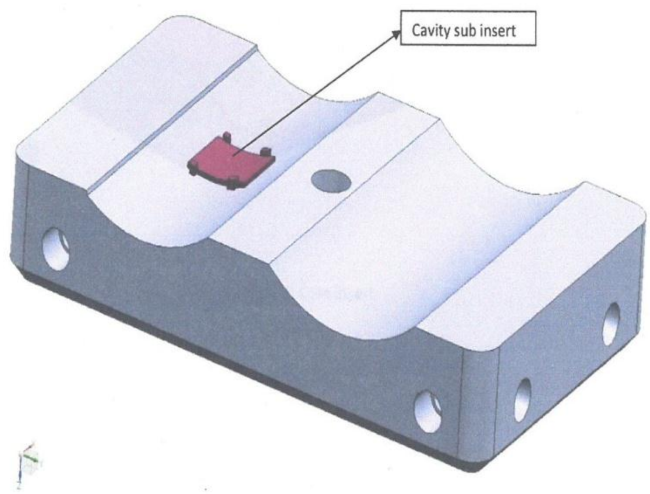


Fig.4 Cavity insert

(b) Core inserts

It is male part of the element in a die which produces a internal surfaces of the mould part and the plastic part will stick to it ejected from the mould. It is fitted at bottom half of the tool. This is rigidly fixed in the core insert plate.

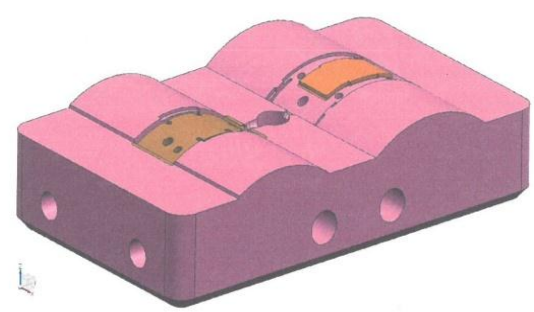


Fig.5 Core insert

2.5.2 Assembly of core and cavity inserts:

When the cavity or core is machined from a large plate or block of steel, or is cast in one piece, and used without bolstering as one as one of the mould plates, 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 characteristic. It is not used as much for multi-impression moulds as there are other factors such as alignment which must be taken into consideration.

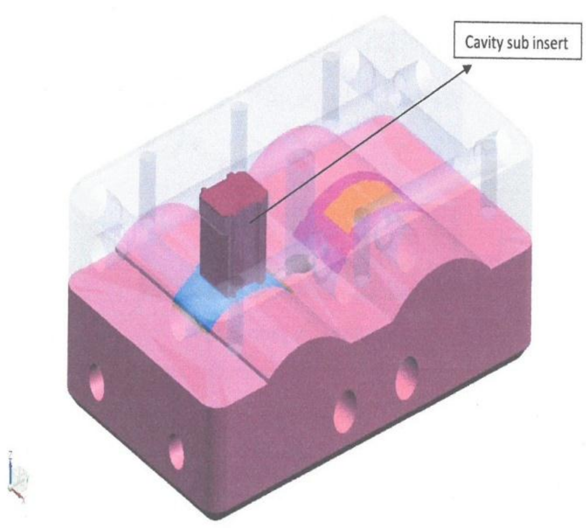


Fig.6 Assembly view of inserts

2.5.3 Mould base:

Mould base is assembly of different plates. The following figures the different plates in 3D

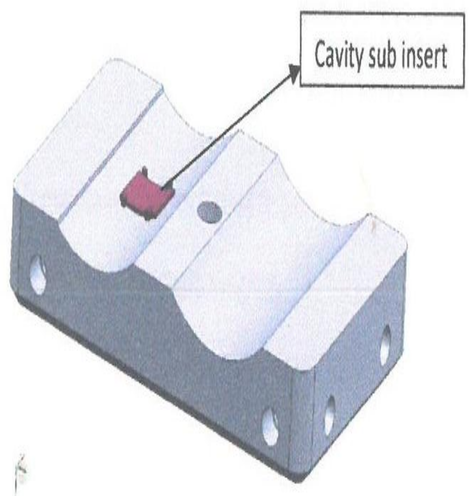


Fig.7 Cavity inserts

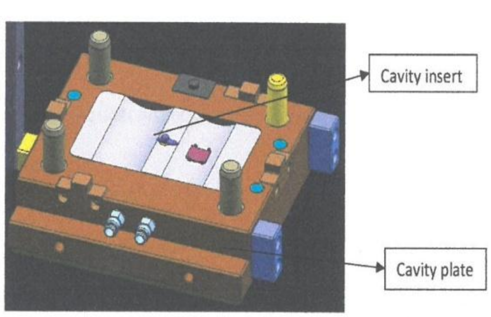


Fig.8 Cavity plate with insert

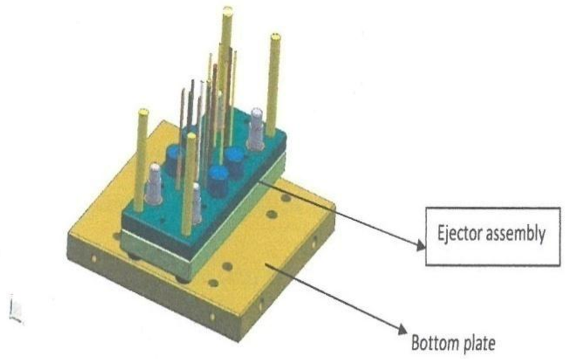


Fig.9 Bottom plate and ejector assembly

2.5.4 Assembly of mould base:

Assembly process planning:

- First of all the assembly and sub-assembly is to be studied the process is planned considering the functional requirement along with fitment of mating parts showing indications and directions.
- The detail record is maintained of each part required for the assembly right from the material received to the final inspection report.
- The details of the process of each part can be obtained from the job cards. While the dimensions with tolerances can be known from the inspection reports.
- The details of part reaching the assembly can be obtained from the bar chart made before starting the actual manufacturing.

Assembly process:

While assembly of all parts and sub units first of all check the following things.

- Study the drawing.
- Check the component thoroughly.
- Collect analyze the mating parts its dimensions.
- Check the de burring if not then de-burr it.
- Before final assembly, check the fault occurring between mating parts.
- Check all the parts of standard die set and plate thickness for further calculation.
- Check all the standard parts which are being used in this tool.
- All the inserts are maintained as per drawing as per drawing for easy fitment.
- Check the all alignments and fitments of all mating parts.
- Identification marks are marked on each part to avoid further confusion after disassembly.

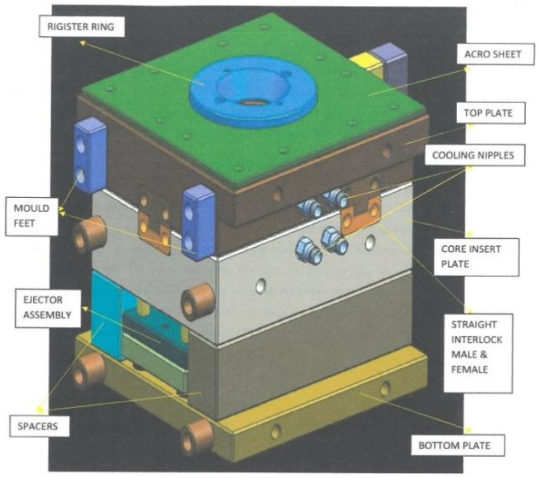


Fig.10 Parts shown in assembly view

Cooling system design

Mould cooling for more than two-thirds of the total cycle time in the production of injection moulded thermoplastic parts. An efficient cooling circuit design reduces the cooling time, which in turn, increase overall productivity. Moreover uniform cooling improves part quality by residual stresses and maintain dimensional accuracy and stability.

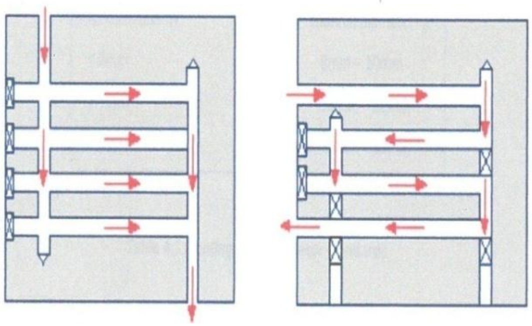


Fig.11 Parallel and series cooling channels

Cooling line configurations

The cooling system in plates and inserts. The cooling water flows one side of the plate and circulate around the inserts and flows from opposite side of the plate. These hot flows into chillers and get cooled up to 12^0c and again re circulates into die.

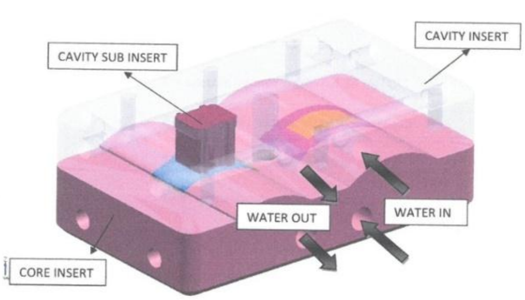


Fig.12 Inserts and plate cooling system

Calculations:

Amount of heat energy generated is Q= m × C × ΔT

Where m= mass of water=1

C= specific heat of water = 4.18 j/g^0C

ΔT= change in temperature of water (T_o-T_i).

T_i= inlet temperature of water

To=outlet temperature of water

Q= 2.5 × 10^4 j/kg

- 2.5 × 10^4 j/kg = 1× 4.8 × 1000 × (T_o-20^0c)

(T_o-20^0c) = 5.94

T_o = 20+5.94

T_o = 25.94^0c

Mould base specifications

W_t = H × W × L × ρ (kgs)

Where

H = height of mould = 246mm

L = length of mould = 246mm

W = width of mould = 223mm

ρ = density of material C45 = 0.000007831kg/mm^3

ρ = density of material MS = 0.00000775kg/mm^3

III. Results and Discussion:

In this paper, Three trails were made in first and second trail some defects were observed in some components like short filling, appearance of parting line bar, air traps etc. remedies were made on trial and error basis, final optimum input values like injection pressure, cooling time nozzle temperature etc, are finalized mass production.

Trail-3 was successfully trailed and the components were obtained without any defects therefore trail three values are fixed for mass production. These components are inspected as per the component drawing.

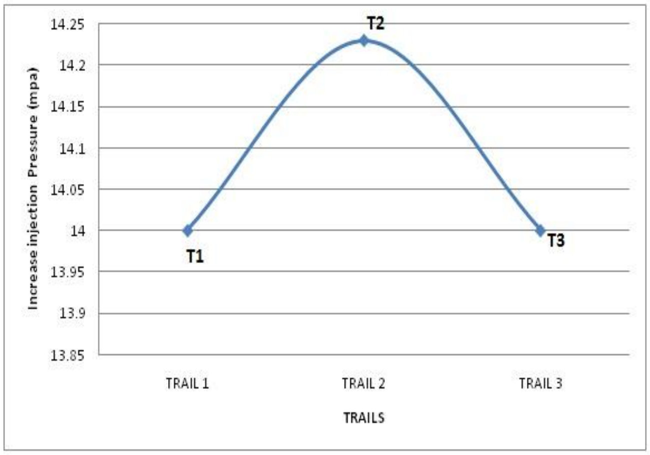


Fig.13 Showing increase in injection pressure

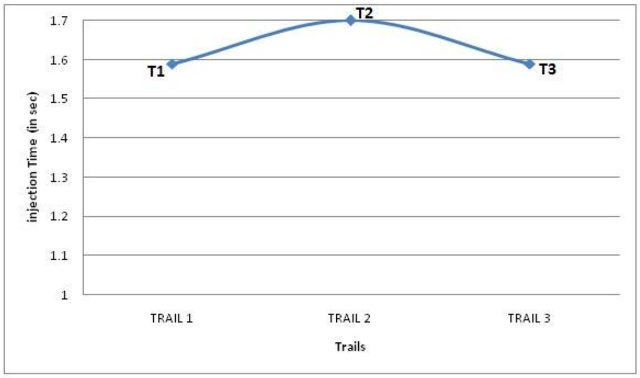


Fig.14 Showing increase in injection time

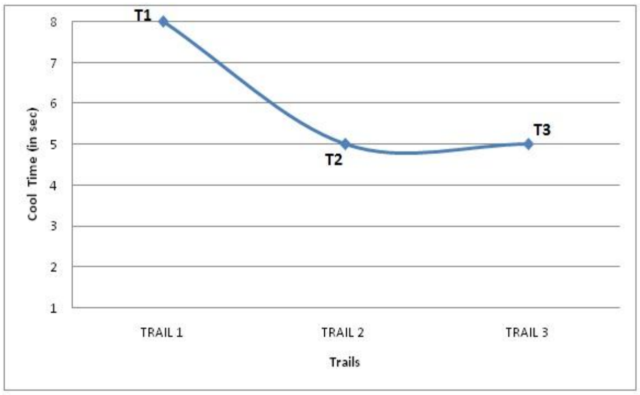


Fig.15 Showing increase in cool time

Advantages of injection moulding:

- Parts can be produced at high production rates without any further works.
- Intricate shapes can be easily moulded.
- Close dimensions tolerances can be easily maintained.
- Complete automation of process is possible.
- No scarp (runners and gates are reused).
- Parts can be moulded with metallic and non metallic inserts.
- The production rate will be more.

Possible issues from the production process:

- Shrinkage
- Weld lines
- Burning
- Wrap
- Poor appearance
- Shrinkage mould
- Rotating mould

IV FUTURE SCOPE

As we done Design and development of family injection mould for Innovative Plastic IN and Innovative Plastic OUT" we can also optimize the design consideration by giving proper inputs to minimize the production cycle time. The cost of the family mould can be reduced. There are a number of future research directions to extend and improve this work.

V CONCLUSION

The multi injection mould for Innovative Plastic IN & Innovate Plastic OUT has been successfully designed and developed according to the customer requirements. In the project we are very clearly show the manufacturing process of mould for "IP IN & Ip OUT' and processing method used for the product. The material selection and cost estimation of the family mould is also calculated. Three trails were made to check the quality of product in first and second trail some defects were observed in some components like short filling, appearing of parting line bar, air traps etc., were observed in few components. Remedies were for those defects on trail and error basis. Finally optimum input values like injection pressure, cooling time nozzle temperature etc, are finalized for mass production.

Referances

1. *R.G.W. Pye, "Injection mould design", eorge godwin, london and newyork, 1983 published by plastics and rubber institute.*
2. *M. Parvin and J.G. Williams (1975). "the effect of temperature on the fracture of poly carbonate".*
3. *J. Bloom, A. Lindemann (2003/2007). "characterization of thermo physical properties of molten polymers and liquids using the flash technique and Hign temperatures-high pressures.*
4. *George godwin. Injection mould design (1993).*
5. *Bryce, douglas M plastic injection moulding: manufacturing process of fundamentals SME, 1996.*
6. *Bryson, plastic materials, butterworth's 9th ed (1999).*