International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 4 Issue 05, May-2015

Design of Singe Impression Injection Mould for Lower Bearing Cover

Vishwanath DC
Student, M. Tech
Government Tool Room and Training Centre
Mysuru, India

Dr. D Ramegowda
principal,
Department of PG Studies
Government Tool Room and Training Centre
Mysore, India

***Abstract*— Injection moulding is one of the techniques used to produce plastic products by forcing the molten plastic material under pressure into a mould, where it is solidified and subsequently released by opening the two halves of the mould. The tool design and development is a specialized and critical area. This paper presents the conceptual design of plastic injection mould. The method represented here is for the design of a two-plate and split-cavity injection mould. The material used for the production of the component is Nylon 66 GF30. The technique is incorporated to produce a good quality component considering the ease of manufacturability and positive ejection of the component within the minimum possible time and cost. Desired product to be manufactured requires mold and tool. So, the tool design should be such that so as to match the machine specifications and should be accurate and economical for successful life of the component or product.**

Keywords*— *Injection mould,Split mold, Gate, Ejection system,

I. INTRODUCTION

Injection moulding is a manufacturing technique for making parts from both thermoplastic and thermosetting plastic materials in production. This process is most typically used for thermoplastic and thermosetting plastic materials which may be successively melted, Molten plastic is injected at high pressure into a mould, which is the inverse of the product's shape. After a product is designed, usually by an industrial designer or an engineer, mould are made by a mould maker (or toolmaker) from metal, usually either steel or aluminium, and precision machined to form the features of the desired part. Injection moulded components are a feature of almost every functional manufactured article in the modern world, from automotive products. Injection moulding is widely used for manufacturing a variety of parts, from the smallest component to entire body panels of cars. This versatile process allows us to produce high quality, simple or complex components on a fully automated basis at high speed with materials that have changed the face of manufacturing technology over the last 50 years. Injection moulding is the most commonly used manufacturing process for the fabrication of plastic parts. A wide range of products are manufactured using injection moulding, which vary greatly in their size, complexity and application. The injection moulding process requires the use of an injection moulding machine, raw plastic material and a mould. It is a manufacturing process wherein plastic polymers that have been compressed into the form of pellets are fed into an injection moulding machine. The plastic is melted in the injection moulding machine and then injected into the mould, where it cools and solidifies. Once after the melt is solidified, the melt opens and the part is ejected. The cost of the final moulded part is a function of design, material and processing expenses. The processing expenses are dependent upon the cost of the injection equipment and may be reduced by either decreasing the cycle time or producing more than one part per cycle using multi-cavity moulds. Using multi-cavity moulds, several part can be moulded simultaneously with no increase in cycle time

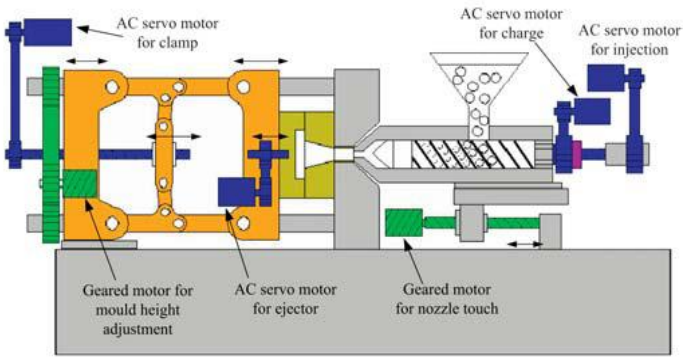

Fig- 1: Injection moulding machine

A. Injection Moulding Cycle

The injection moulding process begins by feeding the plastic material and appropriate additives from the hopper to barrel which is then heated with the sufficient temperature to make it easily flow, then the molten or heated plastic which was melted will be injected under high pressure into the mould this process is commonly known as Injection, After injection pressure will be applied to both platens of the injection moulding machine in order to hold the mould tool together afterwards the product is set to cool which helps it in the solidification process. After the product gets its shape the two platens will move away from each other in order to separate the mould tool which is known as mould opening and finally the moulded product is ejected from the mould. Injection moulding is the most commonly used process to realize plastic parts with high production rates and good control on the product dimensions.

This cyclical process is carried out in three phases:

(i) Filling
(ii) Packing
(iii)Cooling

- Stage 1: Fill stage

During this stage, the mould cavities are filled with molten resin. As the material is forced forward, it passes over a spreader, or torpedo, within the barrel, which causes mixing. This stage is determined by an injection velocity, a pressure, and a time. Injection velocity is the rate at which the plunger moves forward.

- Stage 2: Pack stage

As the melt enters the mould, it cools and introduces shrinkage. The pack stage is necessary to force more melt into the mould to compensate for shrinkage.

- Stage 3: Hold stage

When no more material can be forced into the mould, melt can still leak back through the gate. The hold stage applies forces against the material in the cavity until the gate freezes to prevent leaking of the melt.

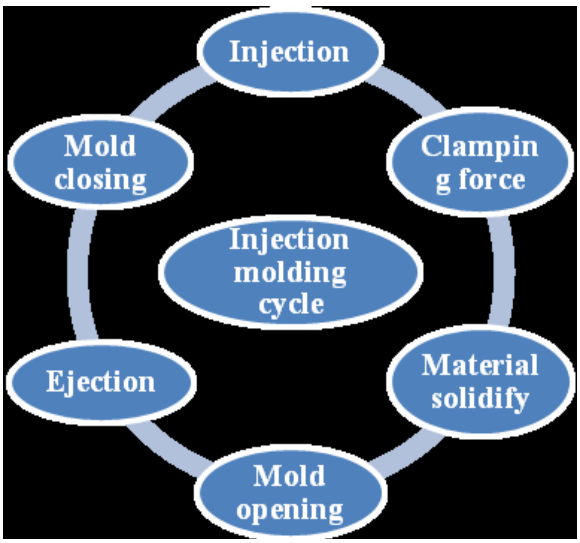

Fig- 2: Injection moulding cycle

B. Basic Design of Injection Mold Tool

Various design considerations had been applied in the design. The mold is designed based on the platen dimension of the plastic injection machine used. There is a limitation of the machine, which is the maximum area of machine platen is given by the distance between two tie bars. Therefore, the maximum width of the mould plate should not exceed this distance. Factors to be considered during designing of injection moulding tool

1. Design and material of components
2. Number of components required
3. Selection of Injection moulding machine
4. Number of cavities
5. Type of tool
6. Selection of parting line
7. Positioning of core and cavity
8. Ejection system
9. Designing of layout
10. Fool proofing arrangements
11. Cooling elements
12. Tool life

C. Two Plate Mould Concept

The two plate mould is simple in design, yet versatile. It consists of a moving and a stationary half. The cavity and core can be mounted on either half , depending on part design and the location of knock-out pins. This mould is easily adapted for different designs and all part ejection methods.

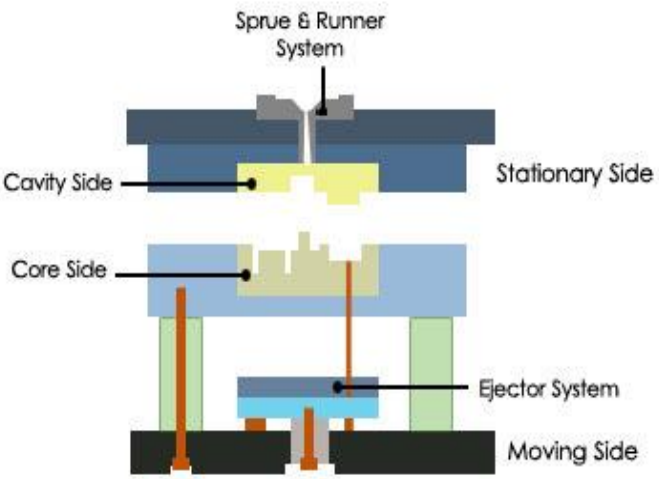

Fig-3 : Standard two plate mould

This consists of two halves fastened to the two platens of the moulding machine's clamping unit. When the clamping unit is opened, the mould halves separate. Mould contains multiple cavities to produce single part in a single shot. The parting surface is the surface shared by the two mould halves. A cooling system is required for the mold. This consists of an external pump connected to passageways in the mold, through which water is circulated to remove heat from the hot plastic. Air must be evacuated from the mould cavity as the polymer rushes in. Much of the air passes through the small ejector pin clearances in the mould. In addition, narrow air vents are often machined into the parting surface.

D. Concept of Split mould

A type of mould consisting of upper and lower half that are compressed together to form a single part

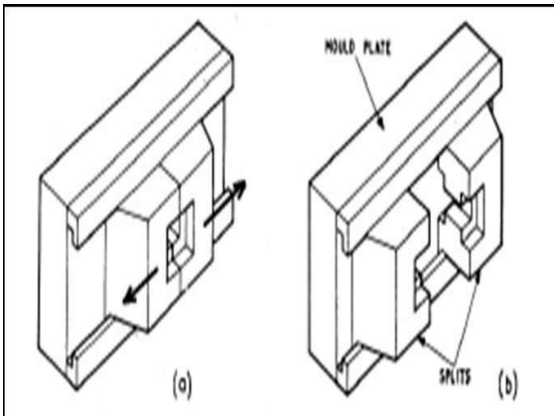

Fig- 4: Basic split mold. Splits shown in Closed (a) and Open (b) positions

- Split moulds are used where there is a significant amount of undercut detail.
- In such cases it is easier to split the mould cavity completely.
- In the case of a split mould the entire cavity separates in order to eject the component.
- This can be achieved using angled guide pins to move the cavity halves sideways as the mould opens.

II. METHODOLOGY

Methodology is the systematic step-by-step planning

a. Study of the component
b. Solid model of the component
c. Step by step design calculation
d. Selection of tooling materials
e. Core and Cavity extraction
f. Solid modeling of the tool
g. Analysis
h. Tool manufacturing process
i. Tool tryout and troubleshooting
j. Cost estimation

A. COMPONENT DETAILS

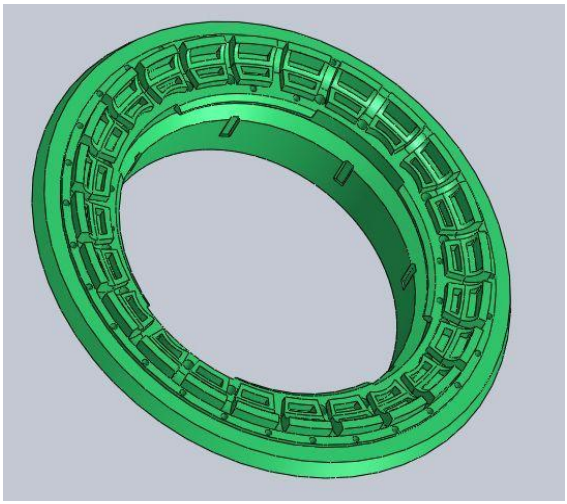

Fig-5: Component 3D model

i. Component Name : lower bearing cover
ii. Component Material : Nylon 66 GF30%
iii. Shrinkage : 0.7%
iv. Component weight : 0.0529 kg
v. Density of the material : 1290 kg/m^3
vi. Volume of the component : 0.000041009m^3
vii. Moulding type : Single cavity injection mould tool
viii. Projected area of the component : 0.006162 m^2 (From CAD model)

Following are the criticality involved in the component –

a. Since this component has an external undercut so split cavity is use.
b. Component has varying diameter along its length and projections at the end for locking grooves purpose. In order to achieve this, movable inserts needs to be used.
c. The tool has to be designed very accurately as the component is to be assembled with other parts.

B. Core and Cavity

Moulds separate into two halves called the core and the cavity as shown in Fig. 4 and 5 to permit the part to be extracted. In general, the shape of a part must not cause it to be locked into the mould.

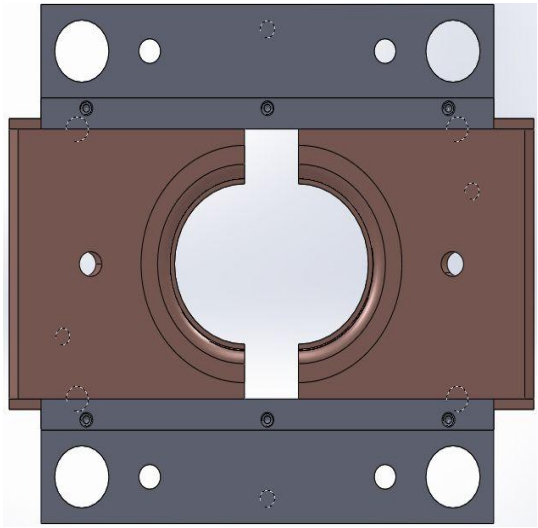

Fig-6: Core

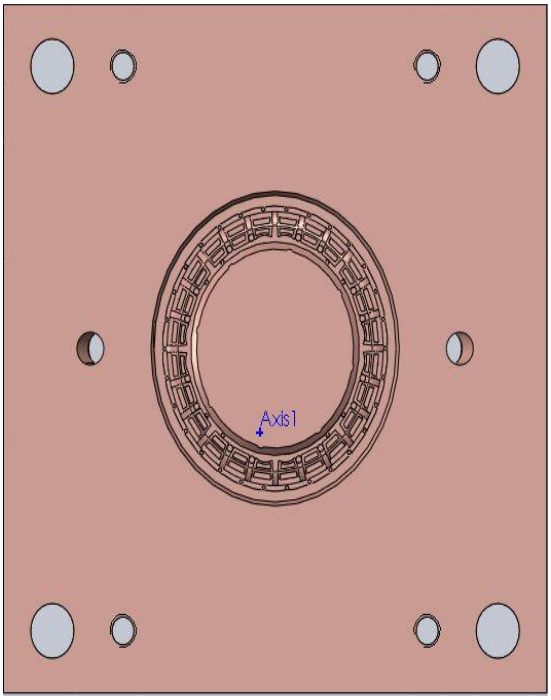

Fig-7: Cavity

B. Feed System

It is necessary to provide a flow-way in the injection mould to connect the nozzle to each impression. This flow way is termed the feed system. Normally the feed system comprises a sprue, runner and gate.

D. Edge gate

This is a general purpose gate and in its simplest from is merely rectangular channel machined in one mould plate to connect the runner to the impression

i. The cross sectional form simple and therefore cheap to machine
ii. Close accuracy in the gate dimension can be achieved
iii. The gate dimension can be easily and quickly modified.
iv. All common moulding materials can be moulded through this type of gate

One disadvantage of this type of gate however, is that after gate removal a witness mark is left on a visible surface of the moulding

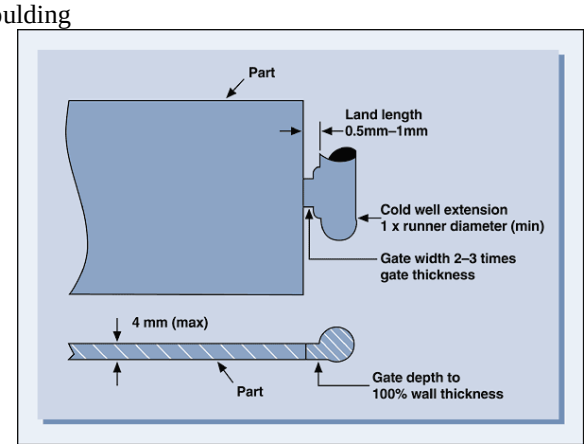

Fig-8 Edge gate

E. Ejection System

An ejection system is needed to eject the molded part from the cavity at the end of the molding cycle. On cooling the moulded component will shrink onto and remain on the core or moving half when the mould opens. This comprises of a:

- Hydraulic actuator
- Ejector plate
- Ejector pins

Here Ejector pins ejection has been employed by going through the component design.

F. Pin ejection system

This is most common type of ejection as, in general it is the simplest to incorporate in a mould. An ejection is very much necessary in order to eject the moulded part from the tool. The design of ejection system is one of the major factors, how efficiently the tool will be in production. Ejector pins built into the moving half of the mould usually accomplish this function. The cavity is divided between the two mould halves in such a way that, the natural shrinkage of the moulding causes the part to stick to moving half. When the mould opens, the ejector pins push the part out of the mould cavity. Pin ejection is used in this tool.

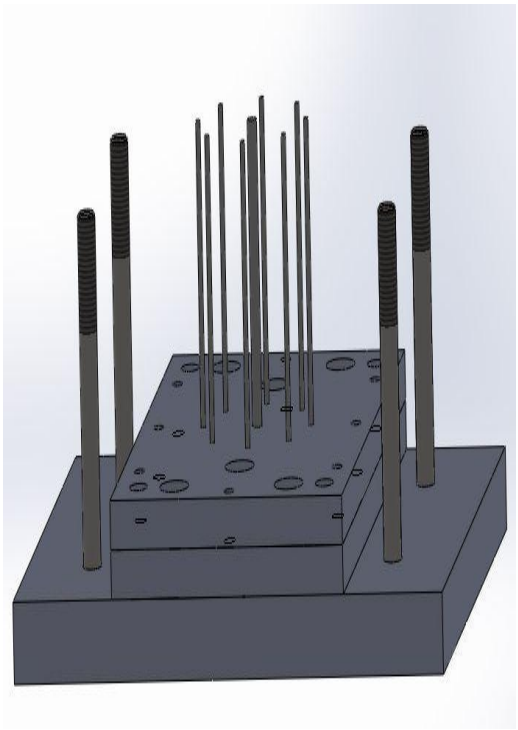

Fig-9 Pin ejection

III. CONCLUSION

The desired component work deals with the concept of designing an injection mould tool for Top bearing cover. The split mould design and pin ejection will make it possible to produce high quality product at minimum cost. The final product is produced with less number of defects involving very little finishing operations.

• REFERENCES

(1) R.G.W Pye, Injection Mould Design, Affiliated East – West Press Pvt Ltd, New Delhi, 4th Edition, 2000
(2) D.V. Rosato, Marlene G. Rosato, Injection Moulding hand book, 3rd Edition, 2012
(3) Peter Jones, the Mould design guide, Smithers Rapra Technology, 2008
(4) S.R. Pattnaik, D.B. Karunaka, P.K. Jha, "Application of Computer Simulation for Finding Optimum Gate Location in Plastic Injection Moulding Process", International Journal of Advanced Engineering Research and Studies, Vol No.1, pp 159-161, March 2012.
(5) Design and Manufacturing of Plastic Injection mold by Dr. Mikó Balázs.
(6) The Mould Design Guide by Peter Zones.