International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 4 Issue 07, July-2015

Conceptual Design of Injection Mould Tool for Top Bearing Cover

Gurutheertha B. G.
Student, Dept. Of PG studies,
Govt. Tool Room & Training Centre,
Mysuru, India

Ramesh Babu K
Faculty, Dept. Of PG studies,
Govt. Tool Room & Training Centre,
Mysuru, India

***Abstract*—This paper presents the conceptual design of plastic injection mould. The method represented here is for the design of a two-plate, four-cavity injection mould. The material used for the production of the component is Nylon 6 GF15. 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. Any product to be manufactured requires machines 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. Solid modeling of the component is done using Solid works software. Various design concepts has been proposed out of which the best one is selected giving valid reasons and applied. Core and cavity, gating system and ejection system of the component has been shown.**

Keywords—Injection mould; Nylon 6 GF15; Bearing cover; Ejection system; Submarine gate

I. INTRODUCTION

One of the most common methods of converting plastics from the raw material form to an article of use is the process of injection moulding. This process is most typically used for thermoplastic materials which may be successively melted, reshaped and cooled. Injection moulded components are a feature of almost every functional manufactured article in the modern world, from automotive products through to food packaging. 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 or so. 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 thermoplastic 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 mould is opened and the part is ejected [1].

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. Also, a mould may contain any number of cavities as long as the required clamp tonnage does not exceed the clamping tonnage of the machine.

In this paper, an attempt has been made to design an injection moulding tool for producing four similar components each having the same material Nylon 6 GF15.

II. INJECTION MOULDING CYCLE

The sequence of events during the injection moulding of a plastic part is called the injection moulding cycle. It is shown in Fig.1. A typical sequence of injection moulding cycle is as follows [2]:

Starting with an empty cylinder, raw material from the feed hopper falls onto the rear flights of the screw which conveys material to the front of the cylinder. During its passage along the cylinder, it is plasticized to a fluid state with the help of external heaters on the barrel. The mould closes and the cylinder moves forward on its carriage until the nozzle is in contact with the entrance of the mould.

The screw is moved forward by the hydraulic cylinder to discharge the thermoplastic material out of the injection barrel through the injection nozzle, and into the moulding tool. Thus, the injection takes place.

The screw is held in the injected position for a given period of time, called as holding time so that a holding pressure may be maintained on the solidifying material within the mould, thus allowing compensating material to enter the mould as the part solidifies and shrinks. As the material solidifies, hold pressure is decayed to zero.

Next is the mould cooling phase wherein time is given for the heat in the moulding to get dissipated into the mould tool such that the moulding temperature falls to a level where it can be ejected from the mould without excessive distortion or shrinkage.

Finally when the cooling phase is complete, the mould is opened and the moulding is ejected.

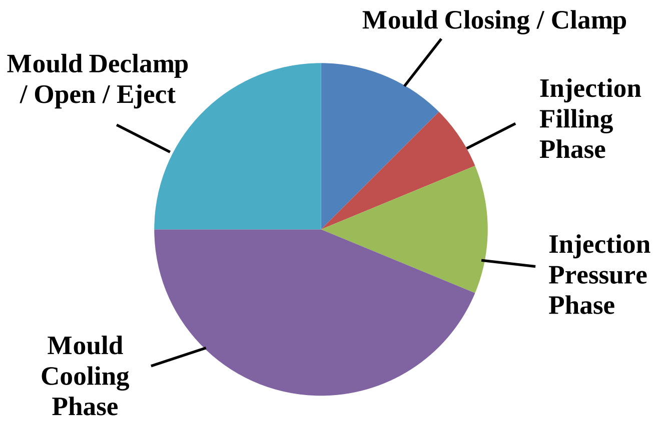

Fig.1: Injection moulding cycle

III. METHODOLOGY

Methodology is the systematic step-by-step planning-

- Solid model of the component: This is done using 3-d modeling software. Here, a 3-dimensional model of the component is prepared considering all the critical dimensions.
- Component details: Various details pertaining to the component such as wall thickness, volume, projected area etc. are collected.
- Concept generation: Here, various concepts are generated for the basic design of an injection mould tool. Concepts are generated by taking into account some key factors. The concepts are evaluated one by one and suitably eliminated until a final concept is left, which is to be used.
- Core and Cavity extraction: Extraction is done by providing proper shrinkage values and tolerance values to the component dimensions. This gives the core and cavity of the component.
- Gating and ejection system: Out of the various types of gates and ejector systems, suitable ones are selected depending upon the component and it is applied.

Fig.2: Component Model

IV. COMPONENT DETAILS

The component is modeled using the software Solid works 2011. The attempt to design a mould tool which can be manufactured and other details of the model is given below.

For the above component, Nylon 6 GF15 is used for the production of the component. It has good dimensional stability which makes it suitable for moulding purpose. Also, it has good wear resistance and as a result of it, can be used for manufacturing bearings.

- Component Name : Top bearing cover
- Component Material : Nylon 6 GF15
- Shrinkage : 0.7%
- Component weight : 7.56 grams
- Density of the material : 1.22 gram/cm^3
- Volume of the component : 6.2 cm^3
- Maximum wall thickness of component : 1.5 mm
- Moulding type : Four cavity injection mould tool
- Projected area of the component : 19.36 cm^2 (From CAD model)

Following are the criticality involved in the component –

- Wall thickness of the component is unequal and very less.
- The tool has to be designed very accurately as the component is to be assembled with other parts.

V. BASIC DESIGN OF INJECTION MOULD TOOL

This section describes the design aspects and other considerations involved in designing the mould. Three design concepts has been considered in designing of the mould including:

1. **Two-plate mould (concept 1)** with single cavity. Since the number of components to be manufactured is more, this is not applicable. This concept is shown in Fig.3.

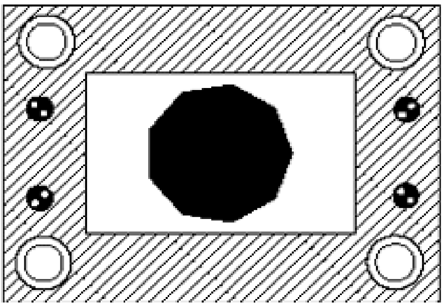

Fig.3: Concept 1 - Single Cavity

2. Two-plate mould (concept 2) with four cavities using edge gate. Edge gate is the most commonly used gate design. This concept is not applicable since this gate has to be manually trimmed using secondary operation which increases the cost of production. Also, wall thickness of the component is very less which is difficult to achieve using edge gate. This concept is shown in Fig.4.

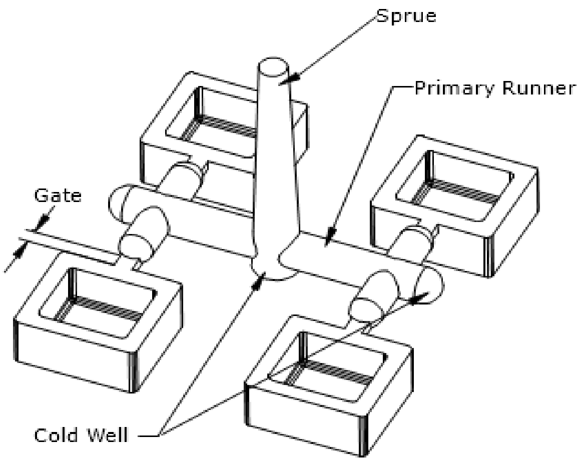

Fig.4: Concept 2 - Four Cavity with Edge gate

3. Two-plate mould (concept 3) with four cavities using submarine gate. This gate is automatically trimmed during the ejection. Also, it minimizes the scars on the component. So, this concept is applicable. This concept is shown in Fig.5.

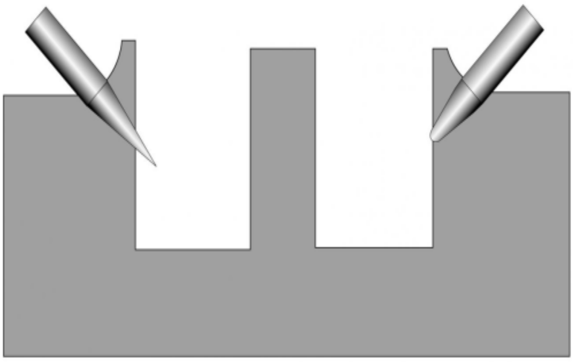

Fig.5: Concept 3 - Four Cavity with Submarine gate

In designing the mould, third design concept has been applied. The mould 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 the machine platen, is given by the distance between the two tie bars i.e. 360 × 360 mm. Therefore, the maximum width of the mould plate should not exceed this distance (360mm).

Factors to be considered during the design of any moulding tool –

- Design and material of components
- Number of components required
- Selection of injection moulding machine
- Number of cavities
- Type of tool
- Selection of parting line
- Positioning of core and cavity
- Ejection system
- Designing of layout
- Fool proofing arrangements
- Cooling elements
- Tool life

All the above factors have to be considered during the designing which affects directly or indirectly in order to obtain greater control over the process and process parameters.

VI. 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 [2]. Fig.6 shows a standard two plate mould.

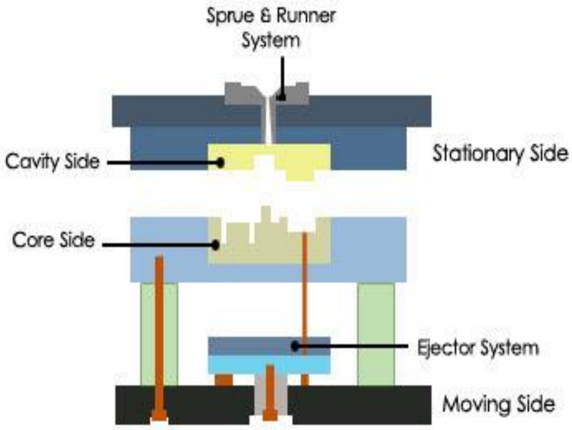

Fig.6: Standard two plate mould

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 mould. This consists of an external pump connected to passageways in the mould, 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.

A. Core and Cavity

Moulds separate into two halves called the core and the cavity as shown in Fig. 7 and 8 to permit the part to be extracted. In general, the shape of a part must not cause it to be locked into the mould [3].

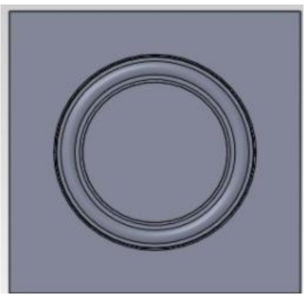

Fig.7: Core

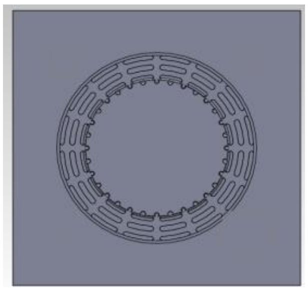

Fig.8: Cavity

B. Gate Type

A submarine gate is used in two-piece mould construction. An angled, tapered tunnel is machined from the end of the runner to the cavity, just below the parting line. As the parts and runners are ejected, the gate is sheared at the part.

Typical gate sizes are 0.8mm to 1.5mm, for glass reinforced materials, sizes could be larger. A variation of the tunnel gate design is the curved tunnel gate where the tunnel is machined in the movable mould half. Auto de-gating is possible by using this gate [4][5]. Submarine gate is shown in Fig.9.

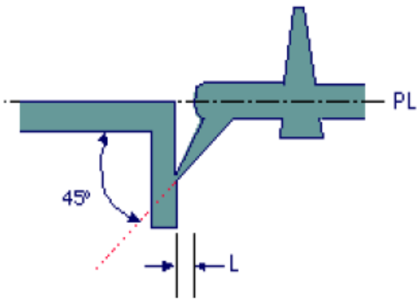

Fig.9: Submarine gate

C. Ejection System

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 [6]. Fig.10 shows pin ejection.

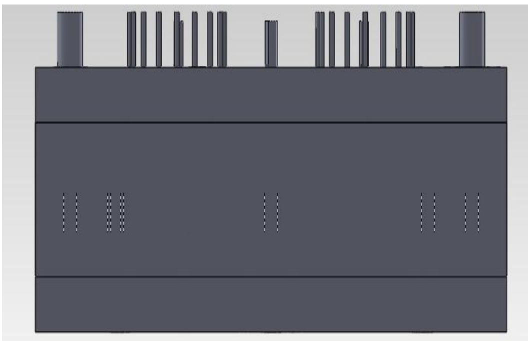

Fig.10: Pin ejection

VII. CONCLUSION

The work deals with the concept of designing an injection mould for Top bearing cover consisting of four cavities. Submarine gating system is used along with pin ejection. The two plate mould design will make it possible to produce a high quality product at a minimum cost. Proper supply of heat and temperature will overcome the effects on the part being produced. The final product is produced with less number of defects involving very little finishing operations.

REFERENCES

[1] Branislav DULEBA and František GREŠKOVIČ, Materials for Injection Molds Production, The 15th International Scientific Conference, Trends and Innovative Approaches in Business Processes,2012 .

[2] Peter Jones, The Mould design guide, Smithers Rapra Technology, 2008.

[3] Haribabu Kummara, Dr. G. Harinath Gowd, Design and Fabrication of Plastic Injection Molding Tool for Pump Gaskets, International Journal of Advanced Engineering Research and Science, Vol 1, Aug 2014, pp 38-45

[4] Vikas B.J, Chandra Kumar R, Influence Of Feeding System In Injection Moulding For Lower Washer Of A Bearing, International Journal of Research in Engineering and Technology,Vol. 02, Aug 2013, pp 396-399

[5] Rahul Vashisht, Arjun Kapila, Analyzing Effects Of Different Gates On Component and Molding Parameters, International Journal of Current Engieering and Technology, Vol 4, Aug 2014, pp 2352-2359

[6] R.G.W Pye, Injection Mould Design, Affiliated East – West Press Pvt Ltd, New Delhi, 4th Edition, 2000