International Journal of Research
eISSN: 2348-6848 & pISSN: 2348-795X Vol-5 Special Issue-13
International Conference on Innovation and Research in Engineering, Science & Technology
Held on 23rd & 24th February 2018, Organized by Tulsiramji Gaikwad Patil College of Engineering & Technology, Nagpur, 441108, Maharastra, India.

Design And Analysis Of Bottle Flip Cover

Amit A. Athawale[1], Professor Vaibhav Bankar[2]

[1]Department of Mechanical Engineering ,Vidarbha Institue Of Technology Nagpur, Maharashtra (INDIA).
Email ID- amuathawale@gmail.com

[2]Head of Department of Mechanical Engineering, Vidarbha Institue Of Technology Nagpur, Maharashtra (INDIA).
Email ID - vhbankar@gmail.com

Abstract*: Injection moulding is a method of forming a plastic product from thermoplastics by feeding the material through the machine component called the hopper to a heated chamber in order to melt it and force the material into the mould by the use of the screw. While producing plastic components using normal/standard multi-cavity mould, we are facing the problems like partial filling, cavities in components, less product quality, injection pressure and temperature drop age and warpage etc. In this paper, the bottle flip cap was redesigned by doing modification in it and due to this it will be beneficial for further use and commercial use also. The shape of previous used bottle cap was modified to another shape and due to this aesthetic view, the material requirement get reduced as compared to the previous one or existing one component. The single thread cap was replaced with multithread cap to make it easy fastening with bottle to save time. For making of bottle flip cap moulding technique or process was used and after that the mould flow analysis of the component was also observed by using solid work and ansys software. The strength of bottle flip cap was also analysed by using software as well as creating design of the component using solid work and ansys software .*

Keywords: Warpage, Moulds, multithread, solidwork, etc

1. Introduction

Injection moulding is a manufacturing process for producing parts by injecting material into a mould. Injection moulding can be performed with a host of materials, including metals, glasses, elastomers, and most commonly thermoplastic and thermosetting polymers. Material for the part is fed into a heated barrel, mixed, and forced into a mould cavity where it cools and hardens to the configuration of the cavity. The manufacturing of thin-wall products is very important for the automotive industry because thinner components allow considerable overall weight savings, beneficial effects on the reduction of fuel consumption and improvement of environmental impact. In addition, the decrease in thickness allows significant cuts in production costs due to less material being used and shorter cycle times. All materials used for automotive applications such as metals, foams, plastics and composites are investigated in order to achieve reductions in product thickness. In particular, thin-wall fabrication of plastic products allows the realization of smaller and lighter parts which can withstand day-to-day use while maintaining their aesthetic appearance.

2. Literature Review

Since 1949, Plastics has specialized in the custom design, tooling, and injection insert moulding of products used in the medical, telecommunications, aerospace and consumer products industries, most plastic materials have greater chemical resistance than most metals. Plastics do not rust or oxidize as metals do and most are not affected, as are metals, by acids or base compounds. The moulding may cause defects and its processing offers a challenge during its development phase. The cost of the mould is high and any process that is not optimized renders heavy overheads during its development cycle and production. So designing the mould which ensures best suitability for the features on the component with smooth flow of molten plastic is very important part of development process. The Plastic adviser

International Journal of Research
eISSN: 2348-6848 & pISSN: 2348-795X Vol-5 Special Issue-13
International Conference on Innovation and Research in Engineering, Science & Technology
Held on 23rd & 24th February 2018, Organized by Tulsiramji Gaikwad Patil College of Engineering & Technology, Nagpur, 441108, Maharastra, India.

Software is one of software used to ensure the best choice location of feeding system and size of gate, runner and sprue. With the aid of mould flow analysis, engineers can obtain statistical data of the moulding process before the mould is actually constructed. The object is to optimize the fill process of a mould and the integrity of the moulded part. The data provided during the analysis helps the engineer select the optimum location for gate, sprue and runner. Temperature variations and all the suitable parameter are clearly defined in the simulation. The end result is accurate, economical and reliable plastic parts. The successful launch of any plastic product depends knowing the true costs and profitability before the job is started.

Injection moulding typically involves large volumes of parts. Small cost overheads per part can be compounded to large cost differences over the life span of the part. Major cost components considered here are material, re-grind and machine costs. Scrap, rejections and regrind costs are also accounted in the cost. Injection moulding is used to create many things such as wire spools, packaging, bottle caps, automotive dashboards, pocket combs, some musical instruments (and parts of them), one-piece chairs and small tables, storage containers, mechanical parts (including gears), and most other plastic products available today. Injection moulding is the most common method of part manufacturing. It is ideal for producing high volumes of the same object.

3. Model Study And Modelling Of Component

3.1Design of component

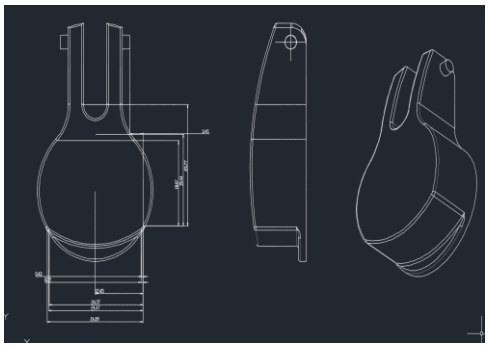

Fig 1: Design of Component

3.2Core and Cavity for making bottle flip cap

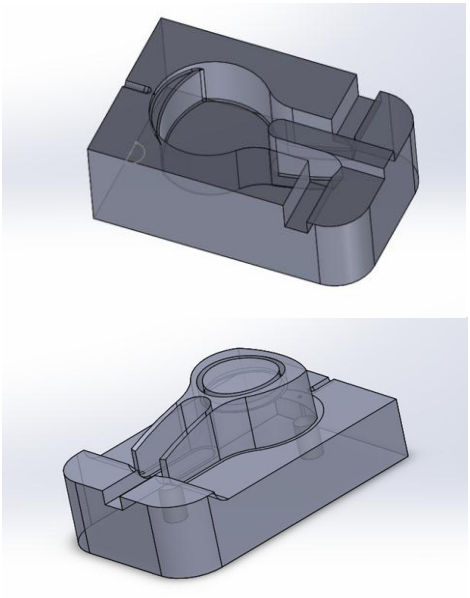

Fig 2: Core and Cavity for making Component

3.3Model of component

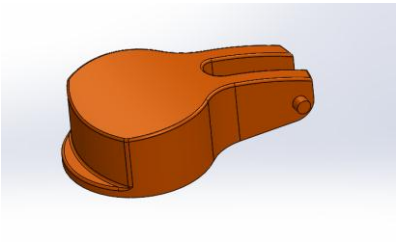

Fig 3: Model of component

4.Design Calculation

4.1 Clamping Tonnage

Clamping tonnage required=Total projected area of (mould) x Cavity pressure

Total projected area of mould flap=61.97cm^2

Injection pressure required for processing polypropylene=1836kg/cm^2

Polypropylene has good flow ability, hence ½ of injection pressure.

Clamping tonnage=total projected area x no. of cavity x 1/2of injection pressure

International Journal of Research
eISSN: 2348-6848 & pISSN: 2348-795X Vol-5 Special Issue-13
International Conference on Innovation and Research in Engineering, Science & Technology
Held on 23rd & 24th February 2018, Organized by Tulsiramji Gaikwad Patil College of Engineering & Technology, Nagpur, 441108, Maharastra, India.

=61.97 x 4 x (1/2 x 1836)

Tonnage required for the component=227553.84 kg

Minimum machine tonnage required=228 Ton

Hence machine used is 228 tonnage capacity

4.2 Plasticizing Capacity

Plasticizing rate of Polypropylene = Plasticizing rate of Ps x (Total heat content of PS ÷ Total heat content of PP)

Where,

Plasticizing rate of Ps = 40kg/hr

Total heat content PS = 239.74KJ/kg

Total heat content PP = 546 KJ/kg

Plasticizing capacity of machine=17.53kg/hr.

4.3 Shot Capacity

SHOT CAPACITY=swept volume x Density of the plastic material x Constant

=100 x 1.2 x 0.95

=114g

Where,

Swept volume = $100cm^3$

Density of the plastic material=1.2

Constant=c=0.35 for crystalline plastics

c=0.95 for crystalline plastics

Shot capacity of machine 114g

4.4 Determination Of Number Of Cavity

According to component shape and size four cavity moulds is preferred

4.5 Wall Thickness Of Core/Cavity Insert

$$\delta = \sqrt[3]{CPd^4/\in y}$$

$$= \sqrt[3]{0.142 X\ 918\ X\ 5.14/2.1\ X\ 10\text{^}6X\ 0.005}$$

=2.05 cm

Where,

C=CONSTANT=0.142

P=CAVITY PRESSURE=918kg/cm^2

D=MAX.DEPTH OF CORE Wall=5.14cm

E=Modulus of elasticity=21.x10^6kg/cm^2

Y=Permissible deflection for the insert=0.005cm

WALL THICKNESS OF CORE/CAVITY INSERT 20.5mm

5. Mould Flow Analysis

It is required to do the mould flow analysis for the particular component to know the proper filling of material into the component. We also know that if any other defects are coming during the filling process of the component by analysis we can remove the defect from the component. Following are some images of analysis.

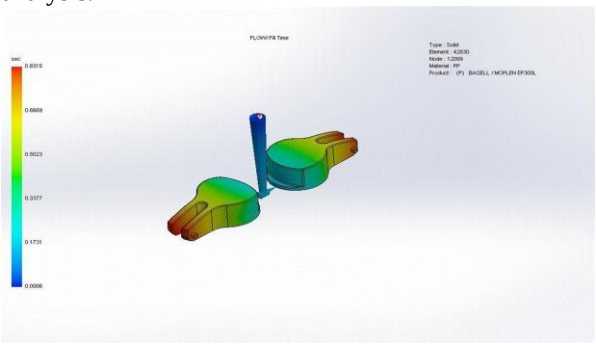

Fig4. Filling of material in component

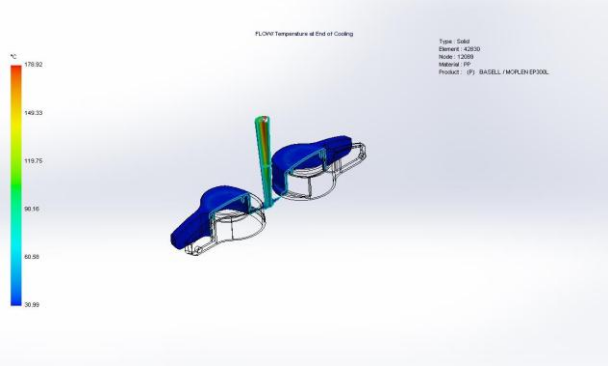

Fig 5.Temperature at end of filling

6. Result

The aesthetic view of the bottle flip cover get changed as well as material requirement also get reduced and due to this cost of the component also get reduced.

7. Conclusion

The complete injection tool is designed for fabricating Bottle flip cover by considering all parameters which are required for the fabrication of component using solidwork. The plastic flow analysis is carried out using solidwork. All the results like fill time, injection pressure, temperature of material, flow rate, quality, etc are analysed.

8. Acknowledgements

I am thankful to Prof. Vaibhav Bankar sir, H. O. D., Department of Mechanical Engineering, for extending their valuable guidance and support. Lastly, I would like to thank and express gratitude towards everybody who at various stages had lent a helping hand.

9. References

i. N.Divya , Dr.Ch.V.S.Parameswara Rao ,Dr.S.S.N.MalleswaraRao " Design and Analysis of Hot Runner Injection Mould for Water Bottle Caps",INTERNATIONAL JOURNAL OF SCIENTIFIC AND RESEARCH PUBLICATIONS, VOLUME 7, ISSUE 3, MARCH 2017

ii. Ms Snehal A. Kamble, A.N.Surde, Swapnil S. KulKarni "Validation for plastic injection mold design for automotive switches", INTERNATIONAL JOURNAL OF ADVANCED ENGINEERING RESEARCH AND STUDIES, OCT-DEC. 2013.

iii. Dr.J. FazlurRahman, Mohammed Yunus, MohammedIrfan, T.M. TajuddinYezdani, "Optimizing the Die DesignParameters for FRP Components Produced in Injection Molding using Mold Flow Analysis", Dr.J.FazlurRahman,Mohammed Yunus, Mohammed Irfan, T.M.TajuddinYezdani International Journal of Engineering Research and Applications (IJERA), (2012).

iv. SharifahImihezri Syed Shaharuddin, Mohd.Sapuan Salit, Edi SyamsZainudin, "A Review of the effect of Moulding Parameters on the Performance of Polymeric Composite Injection Moulding", Turkish J. Eng. Env. Sci. 30, (2006)

v. Bown, J., "Injection Moulding of Plastic Components", McGraw-Hill,

vi. Book on Injection mould design by R.G.W.PYE

vii. http://www.bayer.com/polymers-usa

viii. http://www.academia.edu/1206855/A CADCAE Integrate Injection_Molding_Design System

ix. http://www.compasstech.com/visi-flow/

x. http://en.wikipedia.org/wiki/Injection molding