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 Family Mould for Plastic Box

Mohit Milind Sardare[1], Professor Vaibhav Bankar[2]

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

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

Abstract:-

Injection molding is one of the growing molding process in the plastic field. It is a cyclic process of forming plastic into a desired shape by forcing the material under pressure into a cavity. The shaping is achieved by cooling (thermoplastics) or by a chemical reaction (thermosets). It is one of the most common and versatile operations for mass production of complex plastics parts with excellent dimensional tolerance. Injection molding is preferred where tight tolerance, good quality and high rate productions are required.The aim of this paper is to model, extract core–cavity and develop injection molding tool for manufacturing a plastic box and also to reduce its warpage Core–Cavity design the mould flow analysis of the component is done using SOLIDWORK software.The tool will design to produce a good quality component considering the ease of manufacturability, assembly and Positive ejection of the component. The tool design should match the machine specification and should be accurate and economical for successful life of a component or product.

Keywords: **-** mould, core-cavity, warpage ,etc.

1. LITERATURE SURVEY

Injection moulding is an area where continuous work is being carried out for a long period of time. An attempt has been made to develop a prototype intelligent design system for injection moulds based on usage of internet based technologies as in [3]. Studies have been made for understanding the effect of thermal residual stress and warpage [4].Studies revealed the optimum parameters that minimize the warpage in injection mould using Taguchi approach [5].Researchers had studied cooling channels in the mould and its affect on final product temperature to know the shrinkage rate distribution [6]. Efforts have been made to build a methodology for process selection and manufacturability evaluation of computer based rapid tooling for producing injection moulds [7]. Attempts were made to develop a model so as to have the lowest life cycle cost in the manufacture of injection moulds [8].This paper aims at designing an injection mould tool for fabricating a plastic box and study the parameters by performing the flow analysis on the part. It provides an insight into manufacturability of the mould.

2. INTRODUCTION

Plastic industry is one of the world's fastest growing industries, ranked as one among few billion dollar industries. Almost every product that is used in daily life involves the usage of plastic and most of these products can be produced by plastic injection molding method. Plastic injection molding process is well known manufacturing process to create products with various complex shapes and geometry at low cost .The plastic injection molding process is a cyclic process with four significant stages. These stages are filling, packing, cooling and ejection. The plastic injection moulding process begins with resin feeding and appropriate additives from the hopper to heating/injection system of the plastic injection moulding machine. This is the "filling stage" in which the mould cavity is filled with hot melted polymer at injection temperature. After the cavity is filled, in the "packing stage", additional polymer melt is packed into the cavity at a higher pressure to compensate the expected shrinkage as the polymer solidifies. This is

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.

followed by "cooling stage" where the mould is cooled until the part is sufficiently rigid to be ejected. The last step is the "ejection stage" [2]. In which the mould is opened and the part is ejected, after which the mould is closed again to begin the next cycle. The design and manufacture of injection moulded polymeric parts with desired properties is a costly process dominated by empiricism, including the repeated modification of actual tooling. Among the task of mould design, designing the mould specific supplementary geometry, usually on the core side, is quite complicated by the inclusion of projection and depression [3]. In order to design a mould, many important designing factors must be taken into consideration. These factors are mould size, number of cavity, cavity layouts, runner systems, gating systems, shrinkage and ejection system [3]. In addition to runners and gates, there are many other design issues that must be considered in the design of the molds. Firstly, the mold must allow the molten plastic to flow easily into all of the cavities. Equally important is the removal of the solidified part from the mold, so a draft angle must be applied to the mold walls

2.1 Injection moulding cycle

The sequence of events during the injection mould of a plastic part is called the injection moulding cycle. The cycle begins when the mould closes, followed by the injection of the polymer into the mould cavity. Once the cavity is filled, a holding pressure is maintained to compensate for material shrinkage. In the next step, the screw turns, feeding the next shot to the front screw. This causes the screw to retract as the next shot is prepared. Once the part is sufficiently cool, the mould opens and the part is ejected [1]. Typical cycle times range from 10 to 100 seconds and are controlled by the cooling time of the thermoplastic or curing time of the thermosetting plastic material.

Injection moulding is a cyclic operation shown in fig .1. The cycle consist of

a) Mould close and clamp, (few seconds -depends on machine speeds).
b) Injection - Fill (speed) phase, (few seconds)
c) Switchover and Pack (pressure) phase,(few seconds)
d) Cooling time, (40 to 60% of cycle time)

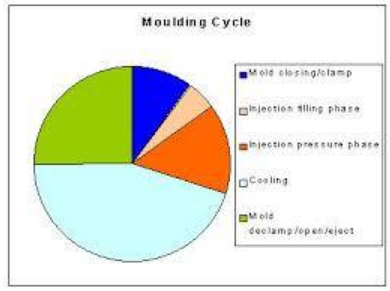

Figure No.1 Moulding Cycle

3. OBJECTIVES OF THE STUDY

The main objective of the study is to design the Injection Mould tool to produce good quality component economically. Also:-

a) To design the family mould
b) .To provide hinge to box and lid.
c) To improve interlocking between box and lid.
d) The study of selected materials has been done, to know its physical and mechanical properties associated with moulding material and moulding characteristics that influence tool design.
e) Apply a shrinkage that corresponds to the part material, geometry and moulding conditions
f) .make conceptual design of mould
g) Design calculations

4. MODEL STUDY AND MODELLING OF COMPONENT

Model study includes identifying the criticality in component, following are the criticality involved in component. Component is modelled using the software solidwork Component has a rectangular structure with dimensions: 125mm (length), 99 mm (width), 37.5 mm (height).Other details of model are given below:-

Component name: plastic box
Component material: PP (polypropylene)
Shrinkage: 1.0-1.1%
Moulding type: single Cavity injection mould tool The injection temperature, time and pressure were 230^0 C, 4.45 Sec and 100.9MPa, respectively are obtained by simulation technique of PP(Poly Propylene)-. Fig. 3 shows the 3D model of box component.

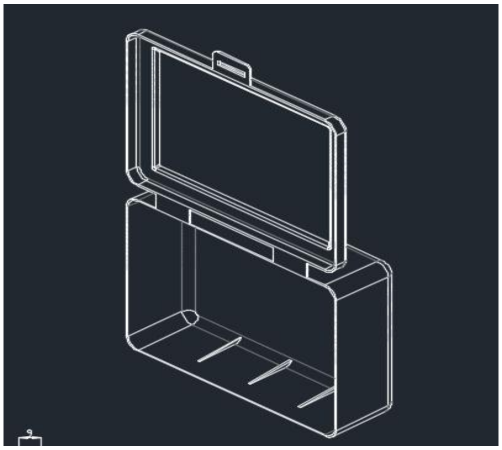

Figure No.2 3D Wire Frame Diagram Of Component

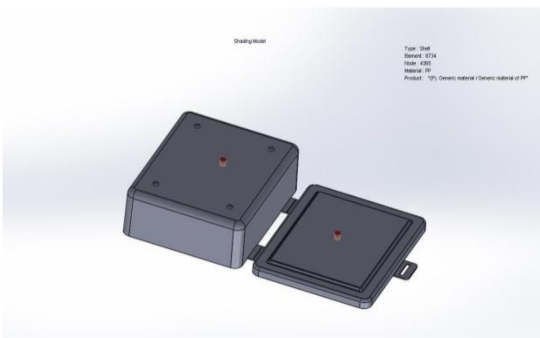

Figure No.3 3D Model Of Component

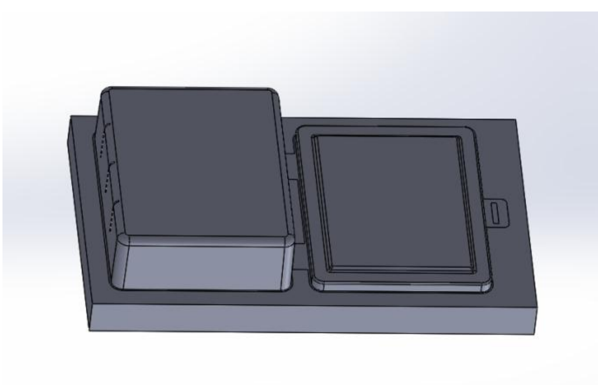

Figure No.4 Core Insert

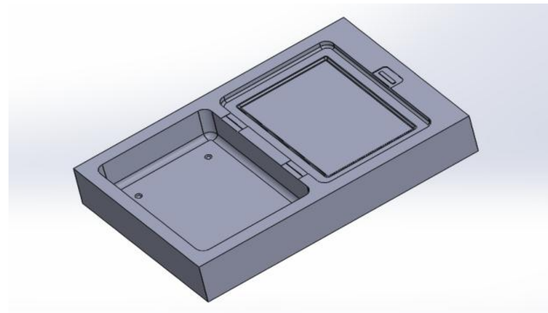

Figure No.5 Cavity Insert

5. DESIGN OF MOULD

Factors to be considered during designing of injection moulding tool

a) Design and material of components
b) Number of components required
c) Selection of Injection moulding machine
d) Number of cavities
e) Type of tool
f) Selection of parting line
g) Positioning of core and cavity
h) Ejection system
i) Designing of layout
j) Fool proofing arrangements
k) Cooling elements
l) Tool life
m) Tool cost

5.1Design calculation

Numeric calculation to be carried out to predict the weight of the component, gate, runner dimension, clamping pressure required, on which machine mold to be loaded, plasticizing and shot capacity of the machine, and cooling parameters like inlet and outlet temperature effect, weight of water to be circulate. these results are compared with the simulation results during moulding.

Actual weight of the component, (W)

W =ρ x V

Where, W =Actual weight of the component in grams,

ρ =Density of plastic material, = 0.946 g / cm3.

V= Volume of the component = 41.21m3

W=0.946 x 41.21

W = 38.98 ≈ 39g.

Clamping tonnage

Clamping tonnage required=Total projected area of (mould) X cavity pressure

Total projected area of mould flap=518

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

Pp has good flow ability, hence ½ of injection pressure

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

=518 x 1 x (1/2 x 1836)

Tonnage required for the component=475524 kg

Minimum machine tonnage required=**475 Ton**

Hence machine used is 475 tonnage capacity

Plasticising capacity

Plasticizing rate of PP= 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.**

Shot capacity

Shot Capacity=swept volume x Density of the plastic material x Constant

=100 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**

Determination of number of cavity

According to component shape and size one cavity moulds is preferred

Wall thicknesses of core/cavity insert

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

$= \sqrt[3]{0.142 \text{X } 918 \text{ X } 1.85^4/2.1 \; X \; 10^{\wedge}6 \text{X } 0.005}$

=0.436 cm

Where,

C=Constant=0.142

P=Cavity Pressure=918kg/cm^2

D=Max.Depth Of Core Wall=1.85cm

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

Y=Permissible deflection for the insert=0.005cm

Wall thickness of core/cavity inserts**4.36mm.**

6. Mould Flow Analysis

It is required to do the mould flow analysis for the particular component to know the proper filling and any other defects coming during the filling process of the component. To locate the proper gatting system and melt temperature of the material in which injection process takes place. Following are some images of analysis.

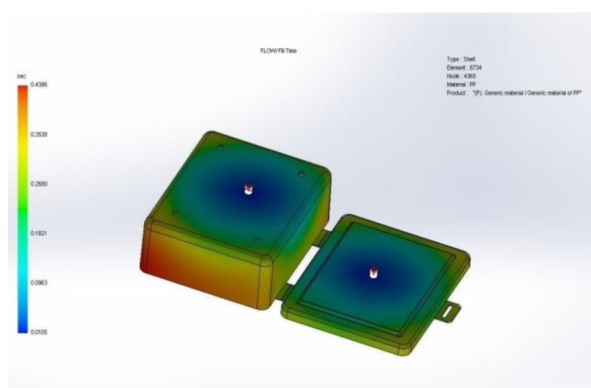

Figure No.6 Analysis Of Filling Time

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.

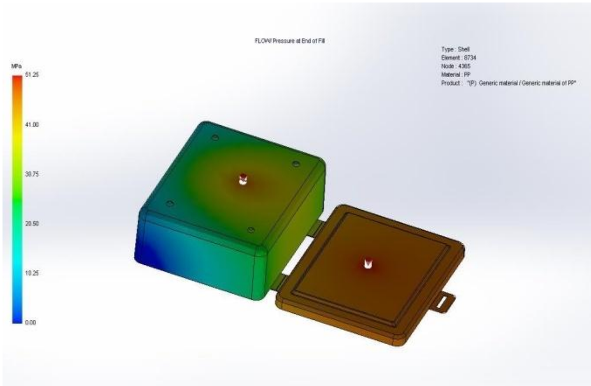

Figure no.7 Analysis of pressure at the end fill

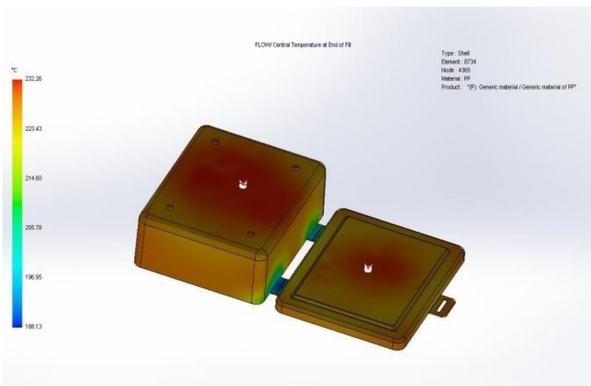

Figure no.9 Analysis of flow central temperature at end of fill

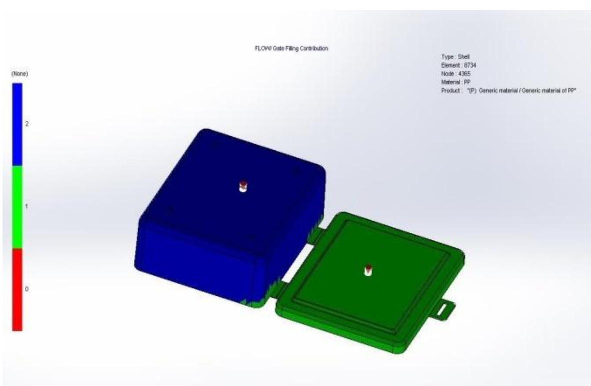

Figure no.10 analysis of flow Gate Filling Contribution

7. Conclusion

The complete injection mould tool is designed for fabricating plastic box by considering the runner design, over flow design, cooling channel design etc. using solidwork.The plastic flow analysis is carried outusing solidwork.All the results viz. fill time, confidence of fill injection pressure, pressure drop, flow front temperature, quality prediction are analysed.

8. Acknowledgements

I would like to take opportunity to thank those who made this Endeavour a success. I would like to express my gratitude to all the staff, friends and family who helped me directly or indirectly in completing this project successfully.

9. References

[1] Madhukumar, Sampathkumar, Nataraj , Kumarswamy , “Design of plastic injection mould tool for air Filter box bottom cover”, INTERNATIONAL JOURNAL OF MODERN TRENDS IN ENGINEERING and research Volume 02, Issue 04, [April – 2015].

[2] Jian ZHOU , Lijun LI , Yihua HU ,Jianguo YANG, Ke CHENG “Plastic mold design of top-cover of out-shell of mouse based on CAE”, INTERNATIONAL JOURNAL OF ADVANCED IN CONTROL ENGINEERING AND INFORMATION SCIENCE, 2011.

[3] C.K. Mok, K.S.Chin, and HongboLan, An Internet-based intelligent design system for injection moulds, Robotics and Computer-Integrated
Manufacturing, Vol. 24, pp. 1–15, 2008.

[4] S.H. Tang , Y.M. Kong, S.M. Sapuan, R. Samin, S. Sulaiman, Design and thermal analysis of plastic injection mould, Journal of Materials
Processing Technology, Vol. 171 pp. 259–267, 2006.

[5] S.H. Tang , Y.J. Tan, S.M. Sapuan, S. Sulaiman, N. Ismail, R. Samin, The use of Taguchi method in the design of plastic injection mould forreducing warpage, Journal of Materials Processing Technology, Vol. 182, pp. 418–426, 2007.

[6] Hamdy Hassan, Nicolas Regnier, Cyril Pujos, Eric Arquis, Guy Defaye, Modeling the effect of cooling system on the shrinkage and
temperature of the polymer by injection molding, Applied Thermal Engineering, Vol. 30, pp. 1547-1557, 2010.

[7] Nagahanumaiah, K. Subburaj, B. Ravi, Computer aided rapid tooling process selection and manufacturability evaluation for injection molddevelopment, Computers in Industry, Vol. 59, pp. 262–276, 2008
.

[8] R.Folgado, P.Pecas, E.Henriques, Life cycle cost for technology selection: A Case study in the manufacturing of injection moulds, Int. J.
Production Economics, Vol. 128, pp. 368–378, 2010.
[9] Mustafa Kurt, O. SabanKamber, Yusuf Kaynak, GurcanAtakok, OguzGirit, Experimental investigation of plastic injection molding:
Assessment of the effects of cavity pressure and mold temperature on the quality of the final products, Materials and Design, Vol. 30, pp. 3217–3224, 2009

[10] AB.IftekharHussain, Mir Safiulla, Mohamed Ali, G.Suresh "Injection Mould Tool Design of Power Box Side Pane" INTERNATIONAL JOURNAL OF INNOVATIVE RESEARCH IN SCIENCE, ENGINEERING AND TECHNOLOGYVol. 3, Issue 2, February 2014.