Design and Analysis of New Family Mould for Plastic Box

Mohit Sardare[1], Vaibhav Bankar[2]
[1]*Department of Mechanical Engineering,Vidarbha Institue of Technology Nagpur, Maharashtra India*
[2] *Professor, Head of Department of Mechanical Engineering, Vidarbha Institue of Technology Nagpur, Maharashtra India*

Abstract- **Injection moulding is a manufacturing process for producing plastic parts from both thermoplastic and thermosetting plastic materials. 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 for successful life of a component or product.**

Index Terms- **mould, core-cavity, warpage, etc.**

I. INTRODUCTION

Injection molding is one of the most important processes in the plastic manufacturing industry. More than one-third of all plastic materials are injection molded, And the mold is one of the main components in the injection molding process. This process is most typically used for thermoplastic materials which can easily be melted, reshaped and cooled. Injection moulded components are a consist 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 brought revolution in manufacturing technology over the last 50 years or so. The present investigation aims at designing an injection mould plastic box and study the parameters by performing the flow analysis on the part. It provides an preception into manufacturability of the mould. It also sketch the methodology for arriving at the unit cost of the product including the tooling costs.

2. LITERATURE SURVEY

Injection moulding is an area where continuous work is going on for a long period of time. An attempt is made to develop a prototype for injection moulds based on usage of internet based technologies as in [1]. Investigation has been made for understanding the effect of warpage [2]. Attempts were also made to develop a model so to have the lowest life cycle cost in the manufacturing of injection moulds [3]. Efforts have also been made to build a methodology for process selection and manufacturing evaluation of computer based rapid tooling for producing injection moulds [4]. Studies show thermal analysis of plastic injection mould,[5]. Researchers show cooling channels in the mould and its effect on final product temperature to know the shrinkage rate distribution [6]. The present studies aims at designing an injection mould tool for manufacturing plastic box and study the Parameters by performing the flow analysis on the model. It provides an insight into manufacturability of the mould.

3. OBJECTIVES

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) Apply a shrinkage that corresponds to the part material, geometry and moulding conditions.
e) Make conceptual design of mould.
f) Design calculations.

3. MODEL OF PLASTIC BOX

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.

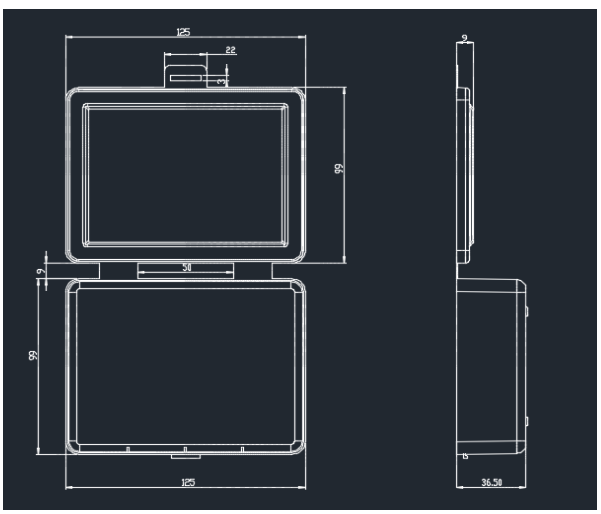

Figure No1 2D Wire Frame Diagram of Component

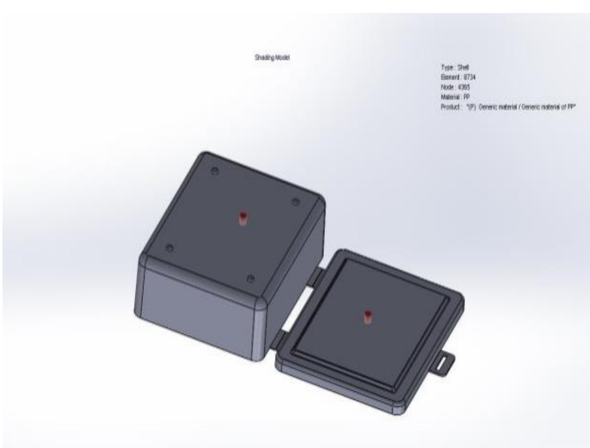

Figure No.2 3D Model of Component

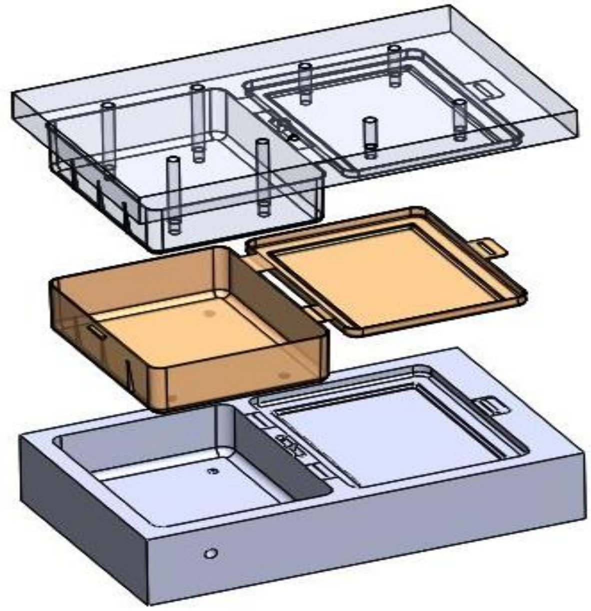

Figure No.3 core and Cavity extraction

5. DESIGN OF MOULD

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=$1836 kg/cm^2$

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 = $100 cm^3$

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

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

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

=0.436 cm

Where,

C=Constant=0.142

P=Cavity Pressure=$918 kg/cm^2$

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.

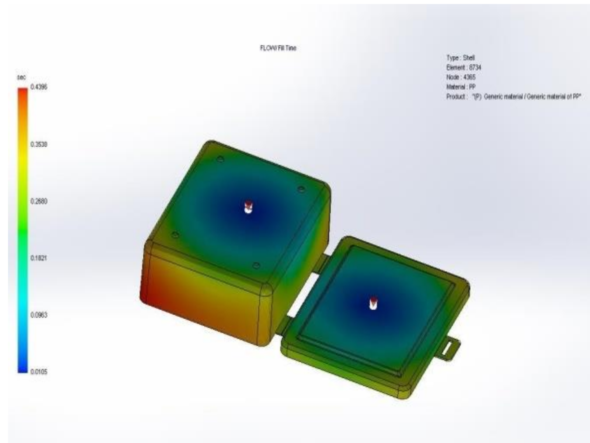

Figure No.4 Analysis Of Filling Time

Figure No.5 Analysis of ease of fill

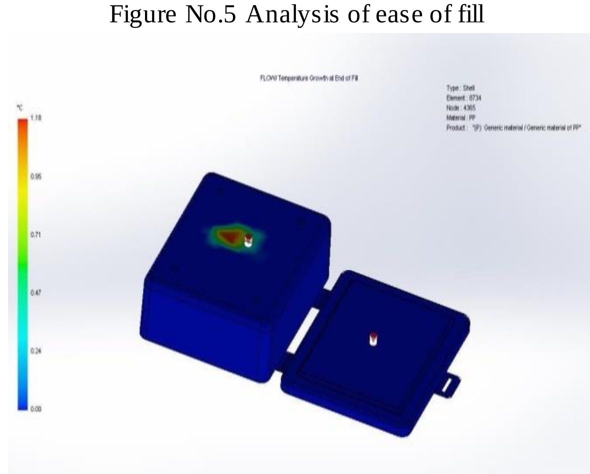

Figure no.6 Analysis of temperature growth at end of fill

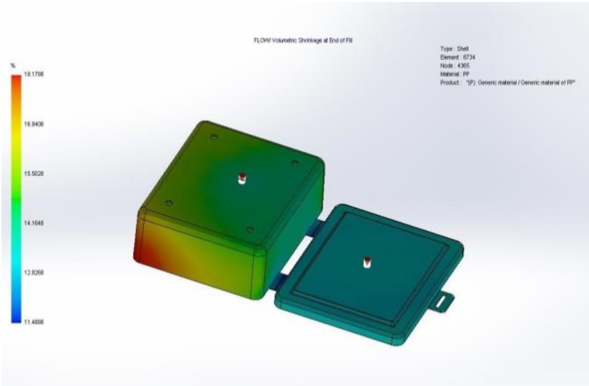

Figure no.7 analysis volumetric shrinkage at the end of fill

7. CONCLUSION

The complete injection mould tool is designed for manufacturing plastic box by considering the runner design, over flow design, cooling channel design etc. using solidwork.The plastic flow analysis is carried out using solidwork. All the results viz. fill time, confidence of fill injection pressure, pressure drop, flow front temperature, quality prediction has been 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 Prof Vaibhav Bankar H.O.D of Department of mechanical engineering and all the staff, friends and family who helped me directly or indirectly in completing this project successfully.

REFERENCES

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

[2] 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 for reducing warpage, Journal of Materials Processing Technology, Vol. 182, pp. 418–426, 2007.

[3] 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.

[4] 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

[5] 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..

[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] Book on All About Plastic By N I George.

[8] Book on Injection mould design by R.G.W.PYE

[9] http://www.academia.edu/1206855/A CADCAE Integrate Injection_Molding_Design System.