IJCRT.ORG ISSN : 2320-2882

INTERNATIONAL JOURNAL OF CREATIVE RESEARCH THOUGHTS (IJCRT)

An International Open Access, Peer-reviewed, Refereed Journal

DESIGN AND ANALYSIS OF FOUR CAVITY INJECTION MOLD FOR COMPONENT BULB HOLDER

[1]Santosh Mudakappa Nevani, [2]Dr.Nagaraja,

[1]Student, M.Tech in Tool Engineering, GT&TC,Bengaluru,India
[2]Principal,Department of PD/PG studies,GT&TC , Bengaluru,India

Abstract: Injection molding is manufacturing process method where the parts are produced by injecting the molten material into the mold. This molding technique, which plays a significant role in the field of plastic processing, is appropriate for the mass manufacture of goods with complex geometries. The component Bulb Holder, which is utilized in homes and other electric devices like lamps and other lighting, is designed and analyzed in this work. This dissertation uses a two-plate injection mould with four cavities because they are more efficient than single cavity injection mould. Due to the small wall thickness of the component ABS material is selected and analysis is done with help of Solidworks 2021, to check the Fill time, cooling time, volumetric shrinkage which helps to analyze before going for production.

Index Terms - **Multi cavity, two plate Injection molding full round runner, edge gate, Solidworks plastics**.

I. INTRODUCTION

Injection Molding is the most widely used for the formation of intricate plastic parts with excellent dimensional accuracy. A large number of items associated with our daily life are produced by exploring injection molding, Typical product categories include house-wares, toys, automotive parts, electronic parts, rigid packaging items, medical disposable syringes and in power tools.

The design and analysis of a two-plate injection mould with multi cavities here we taken four cavities for the component "Bulb holder," weighing 4.05gms, are covered in this dissertation work. The mould is intended to be very productive. The main requirements are a new tool with increased productivity and high-quality components.

2. PROBLEM STATEMENT

To design and analysis the multi-cavity injection mould tool for the component Bulb holder. Which is use in the house hold appliance Solidworks 2021 software is used for both design and analysis. The design of component is tough part in the dissertation because of complex geometry shape. And analysis is done to find the filling time, cooling time and volumetric shrinkage to for check the time taken for filling and cooling to avoid defects on cavity.

3. OBJECTIVES

- Study of component has been made carefully as an existing Two-plate injection mould for four cavities with edge gate was producing components with defects.
- The study of component has been done to design the tool as simplest as possible by using solidworks 2021 software.
- The study of specified material has been done, to know its physical and mechanical properties associated with molding material and molding characteristics that influence tool design.
- Mold flow analysis has been carried out for the component in order to achieve good quality mold before molding by using solidworks 2021.

4. COMPONENT DETAILS

The Component bulb holder is manufactured by using material ABS (Acrylonitrile Butadiene Styrene) having 58mm length, with wall thickness 1mm with 6 ribs of 5mm length at bottom.The 3D model of the component Bulb Holder shown in fig 1.

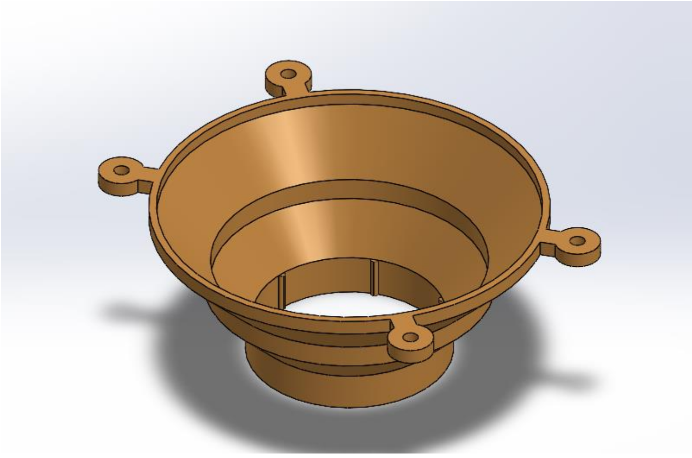

Figure 1 Bulb Holder

Name	Bulb Holder
Material	ABS (Acrylonitrile Butadiene Styrene)
Density	1.01 g/cc
Thickness	1.0 mm
Shrinkage	0.5%
Production	10,000 per month
Volume	$4120.94mm^3$

Table 1 Component details

The Bulb Holder is used in electric lamps/ bulbs to hold the electric bulb. Which is used in everywhere like Household, electric lamps, industry etc. the material was selected for this ABS (Acrylonitrile Butadiene Styrene) because of it having physical properties like good insulating properties and high rigidity etc.

1. TOOL DESIGN

1.1. STAGE 1: STUDY OF COMPONENT

The component Bulb Holder dimensions like size, shape and various cross-section are checked thoroughly by using reverse engineering method with the help of engineering measurement tools like Digital Vernier calliper, radius gauge, micro gauge, bevel protector.

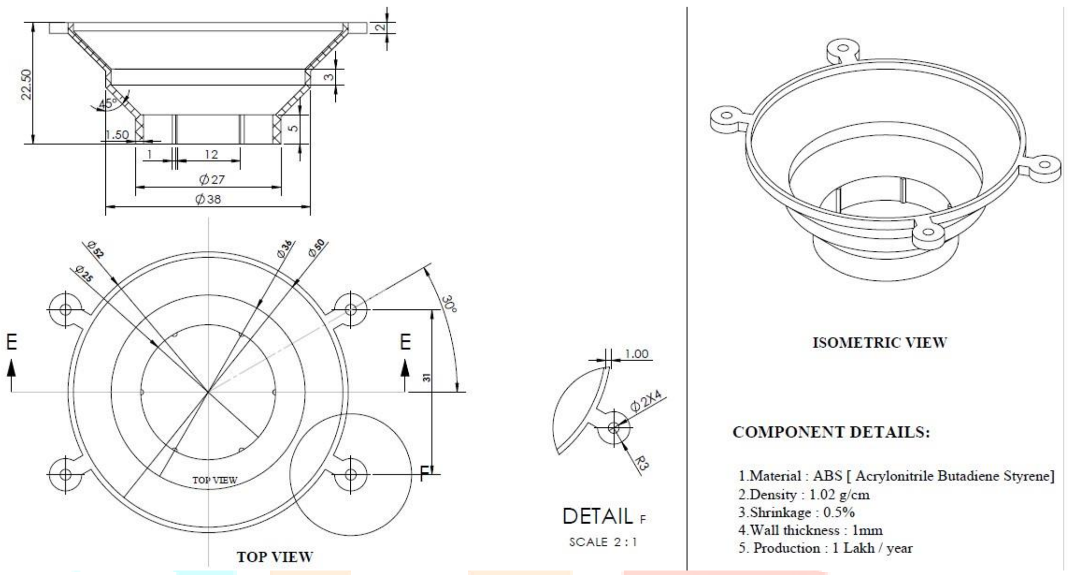

Figure 2 2D Drawing of Bulb Holder

5.2 Design calculation: injection mold of four cavities set based on experience, as applied to "Bulb HOLDER" and various design calculation are given below.

5.2.1 Weight of component (Wc):

Wc = Volume × Density

$$Wc = V_c \times \rho \quad \text{……………………………………………………. (equation 1)}$$

Wc = 4120.94 x 1.01

Wc = 4.08 ≈ 4 grams

5.2.2 Total weight of component

Total weight of component= Actual weight of component × Multifaction factor

Wm = Wc × mf ……………………………………………………. (equation 2)

Wm = 4 x 1.25

Wm = 5 grams/cavity.

∴ For four cavity = 5 × 4 = 20 grams

m_f = Multiplication factor is taken from the table according to weight of the component.

Table showing various values for multiplication factor.

Weight of the Component (grams)	Multiplication Factor(m_f)
0.3-0.5	1.5
1.0-3.0	1.4
3.0-5.0	1.3
5.0-10.0	1.25
10-20	1.10
Above 20	1.05

5.3 Selection of injection molding machine based on clamping force by using following steps:

1. Clamping capacity
2. Shot capacity
3. Plasticizing capacity
4. Number of cavities

1. **Clamping force (C_f)**

C_f = [Projected area of component 20 to 30%] × [1/2 to1/3 of specificinjection pressure] × [Number of cavities] ……………………… **(equation 3)**

PA= πr^2 = 3.142 × 66 = 13,686 mm^2

Injected pressure ABS: - 420-2100 kg/cm^2

=1/2 of max pressure =1/2 ×2100=1050 kg/cm^2

=1/3 of max pressure =1/3×2100=700 kg/cm^2

Considering average, injected pressure = **875 kg/cm^2**

$$C_f = \frac{13.686 \times 30}{100} \times 875 \times 4$$

C_f = 62.3 kgf or Tonnes

∴longer tie bar lengths than other machines, the **SP130** machine type from the SP series is chosen for its tonnage of 62.3T.

SPECIFICATIONS OF SP130 SERIES DGP WINSOR MACHINE

MACHINE TYPE	UNITS	SPECIFICATIONS (SP 130)	
SCREW DIAMETER	mm	45	50
INJECTION PRESSURE	bar	2200	1800
STROKE VOLUME	Cm^3	120	185
SCREW STROKE	mm	190	
MAX INJECTED WEIGHT	gms	270	338
INJECTION RATE	cc/s	169	208
PLASTICISING RATE	g/s	11.1	13.3
SCREW L/D RATIO	–	20	18
SCREW SPEED (MAX)	rpm	165	
HEATING CAPACITY	kW	12.7	
NO. OF HEATING ZONE	–	5	
NOZZLE HOLDING FORCE	kN	60	

2. **Shot capacity for molding machine: -**

Shot capacity = Stroke volume × Density × Plastic material constant ………. **(equation 4)**

Stroke volume = 120 - 185= 152.5 cm^3

Plastic material constant – Amorphous (0.93)

Shot capacity = 337.5×0.93×1.02

Shot capacity =144.661grams

3. Plasticizing capacity: -

Plasticizing capacity (material B) = Plasticizing capacity (material A) $\times \frac{\text{Total heat content of A}}{\text{Total heat content of B}}$

Where,

Material A = PS

Material B = ABS

$\mathbf{Ps_B = Ps_A \times \frac{Q_A}{Q_B}}$ **in Kg/hr ……………………….. (equation 5)**

= 12.2× (57/72)

= 9.658 grams/sec

Refer for IMM and SP130 = Plasticizing rate(g/s)

= 9.658*3600/1000

= 34.76 kg/hr

4. **Number of cavities: -**

a) determined by shot capacity

$\mathbf{Nc = \frac{85\% \times shot\ capacity}{Actual\ weight\ of\ moulding}}$ **…………………… (equation 6)**

Nc = 0.85*(144.661/5)

Nc = 24 Nos

b) determined by plasticizing capacity

Number of cavities = 85%×ps×tc / wm………………… (equation 7)

Where, p- plasticizing rate

W_m-calculated weight of molding per component

T_C-cycle time-assume 15 to 20 sec

Nc =0.85× $\left(\frac{34.76\times20}{3600\times19}\right)$

Nc = **6.2217 ≈ 7 cavities**

c) determined by clamping capacity

Number of cavities (N_c) = 85%×Cf / Pa×Ip……………………… (equation 8)

N_c=0.85(130×1000/13.686×875)

N_c = 4 cavity

∴Select the min 4 cavity for the given exercise by considering the clamping force.

5.4 To calculate wall thickness of core and cavity plate:

$$\textbf{wall thickness of insert}(t) = \sqrt[3]{\frac{\frac{L}{d}\times P\times d^4}{E\times y}} \text{ .... (equation 9)}$$

Where, P-Injection pressure

d- depth of component

E-Modulus of elasticity

y-Maximum deflection

wall thickness of insert(t) =

$$(t) = \sqrt[3]{\frac{0.134\times 875\times 2^4}{2.1\times 10^5\times 0.005}}$$

t = 52mm

5.5 Calculate feed system:

Design of runner: the runner design is calculated by following steps

To calculate runner diameter

d = $d = \frac{\sqrt{w}\times\sqrt{l}}{3.7}$ .... **(equation 11)**

where,

d- diameter of runner

w-weight of molding per component

l-length of runner = 3 to 5mm

$$d = \frac{\sqrt{19}\times\sqrt{40}}{3.7}$$

d=2.97mm ≈ **3mm**

To calculate width of gate(**w**) = $\frac{n\times\sqrt{A}}{30}$..................... **(equation 12)**

(w) = $\frac{0.6\times\sqrt{13,686}}{30}$ = **1.5mm**

Depth of gate **(H) = n×t**........................ (**equation 13)**

(H) = 0.6 × 1 = 0.6mm

6. **FEM ANALYSIS:**

In this FEM analysis, analysis is carried out to find the fill time,cooling time and volumetric shrinkage because to check cycle time to ensure the higher production and avoid defects on the component. The analysis is performed to minimize practical problems and trails. Here,solidworks 2021 software is used for FEM analysis.

- **FILLING TIME**

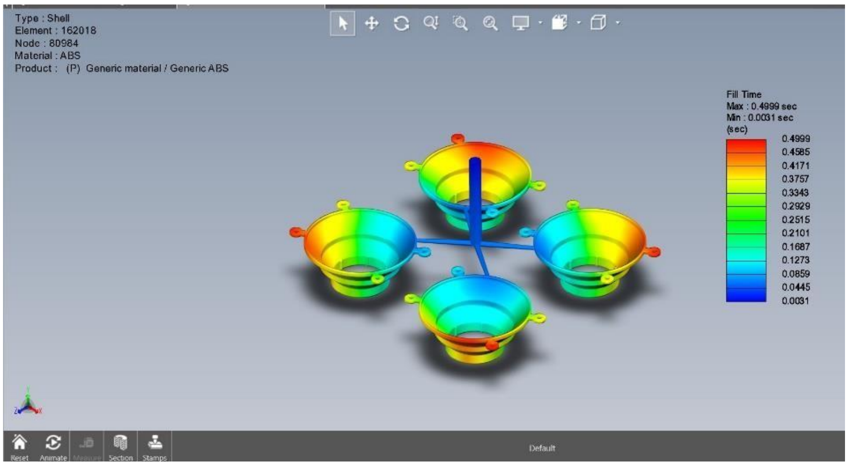

Figure 1 filling time

1. Filling is the initial step in the injection moulding, fill time is the amount of time taken by the mold to entirely fill the core and cavity.

2. Theoretically shorter the fill time higher the production.

3.For this component the fill time which we observed is 0.5seconds to fill cavity.

4.As figure shows red colour indicates the cavity fills late because the part is far from the component.

- **COOLING TIME**

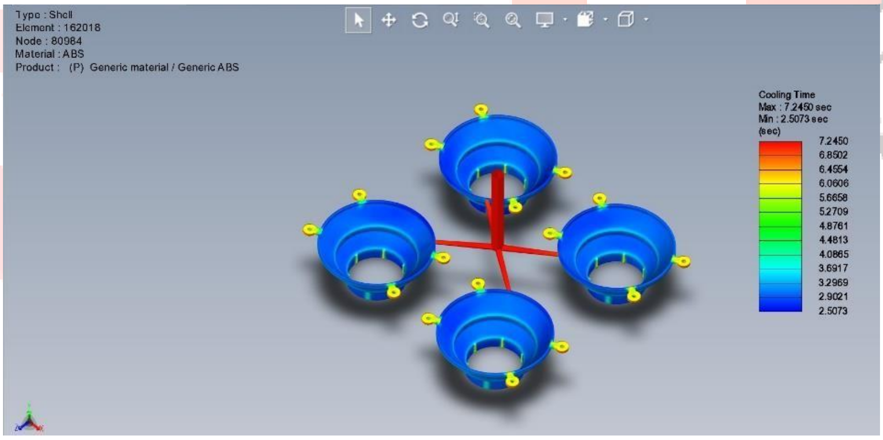

Figure 2 Cooling time

1. Cooling is a very important step in injection molding.
2. In total cycle time nearly 50-80% of time consumes for cooling
3. As above figure shows hottest material is enters from gate and coolest material at end part of the component.
4. From this analysis we observed that cooling time taken for this study is 7.25 seconds.

➢ **Volumetric shrinkage**

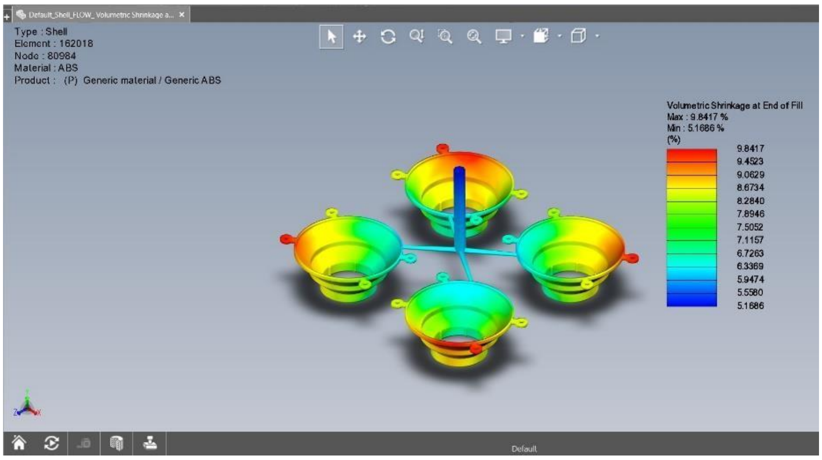

Figure 3 Volumetric shrinkage

1. The volume change of any small amount of material the part from injection through ejection as liquid to solid material is called as volumetric shrinkage.
2. Area shrinkage is the result of volumetric shrinkage along the x, y, and z axes of the component.
3. Volumetric shrinkage for plastic materials typically ranges between 20 and 25 percent throughout the injection molding process.
4. From the above volumetric analysis, it shows that minimum 5.166 % to maximum 9.84%.

6.1 Analysis Results:

Parameter	Result(seconds)
1. Total Cycle time	28
2. Cooling time	7.25
3. Filling time	0.5
4. Holding time	2.6
5. Volumetric shrinkage	5.2% to 9.1%

6. Conclusion:

The component bulb holder was thoroughly designed and analyzed by using injection molding design standards. Study of component has been made carefully as an existing Two-plate injection mould for multi cavities with edge gate was producing components without defects. Solidworks 2021 was used to design for all mould parts and assembly and mould flow analysis like fill time, cooling time and volumetric shrinkage is done, Mold flow analysis has been carried out for the component in order toachieve good quality mold before molding by using solidworks 2021.

REFERENCE

1. Injection molding design, R.G.W Pye, 4th Edition, Affiliated East-West Press Pvt Ltd.
2. Mold Engineering by Herbert Rees, Hanser Publications, New York (2002)
3. WESTERMANN TABLES - HERMANN JUIZ & Edward Scharkus Newage - International Ltd, Publishers.
4. Arpit Arora, Arpit Pathak,Akshat Juneja “Design and analysis of multi cavity injection mould using solidworks” International Conference on Design and Materials (ICDM), 2022.
5. M.V.Kavade,S Kadam, “Parameter optimization of Injection mouldung of ABS by using TAGUCHI Methodology”, IOSR Journal of Mechanical and Civil Engineering (2012).
6. Jayakumar K and Abhishek Basappa, “Design Of Multi Cavity Injection Molding Tool For The Component Torch Body By Using Hydraulic Side core” International Journal of Modern Trends in Engineering and Research (IJMTER) (October 2017).

7. Sen Yeu Yang, Tzu Chien Huang, Po Hsun Huang, Tai Yu Ko "Study on flow imbalance during filling a multi cavity mold using H-type Runners",key engineering materials(January 2008).
8. . Beaunomt, J. Ralston, A. Shuttleworth and M. Camovale, "Troubleshooting cavity to cavity various in multi cavity injection molds", Journal of Injection Molding technology, (1999).
9. J. W. Bozzelli, "Some Reasons Not To Use Multi-Cavity (>4) Tools", ANTEC 2000.
10. G. Y. Su, H. Yokoi, W. M. Yang, "Study on Filling Balance of Plastic Injection Molding in Multi-Cavity Mold with "H" Pattern Runner System", (2003)