International Journal of Advance Research in Science and Engineering
Volume No.07, Special Issue No.03, February 2018
www.ijarse.com
IJARSE
ISSN: 2319-8354

DESIGN & MANUFACTURING OF PLASTIC INJECTION MOULD – THREAD COVER

Shubham Boralkar[1], Pranit Patil[2], Akash Patil[3], Ajay Warekar[4], Rohit Patil[5]

[1,2,3,4,5]Mechanical Engineering,

Nanasaheb Mahadik College of Engineering, Peth, (India)

ABSTRACT

The paper consists of designing a two plate injection mold tool for thread cover. This project deals with, the solution of problem occurred, to protect the threads of shaft. To reduce the wearing of threads of shaft thread cover is used. Solid modeling of mould is create on CATIA V5 R20. VMC programming for mould. There are several methods for plastic moulding. But the injection moulding process is commonly used manufacturing process for the production of plastic parts. The plastic being melted in injection molding machine and then injected into the mould. The material selection for mold design is taken as Mild Steel. Material selection for thread cover is polypropylene. Design and assembly of injection molding die of thread cover. The injection molding die is designed by applying proper design procedure.

Keywords: CATIA V5 R20, Clamping force, Injection moulding, Mould, Plasticizing rate, Shot Capacity, Thread Cover

I. INTRODUCTION

The injection molding process is most commonly used manufacturing process for production of plastic parts. Injection molding is a manufacturing technique for making parts from both thermoplastic and thermosetting plastic materials in production. Molten plastic is injected at high pressure into a mold, which is the inverse of the product's shape. Raw material is melted in the injection molding machine and then forwarded into the mold with the help of reciprocating screw where it cools and solidifies into the final product part. The reciprocating screw is mounted on gear box in the barrel; barrel contains heaters for heating the raw material. In the injection process raw material is melted by heat and pressure. While forwarding material the material enters the grooves of reciprocating screw. The reciprocating screw completes the shot volume and returns to reverse position. The basic injection cycle is as follows: Mold close – injection carriage forward – inject plastic – metering – carriage retract – mold open – eject part. Molds are made by a mold maker (or toolmaker) from metal, usually either steel or aluminum and precision machined to form the features of the desired part. Injection molding is widely used for manufacturing a variety of parts, from the smallest component to entire body panels of cars. Injection molding is the most common method of production, with some commonly made items including bottle caps and outdoor furniture. Injection molding typically is capable of tolerances equivalent to an IT Grade of about 9–14. The most commonly used thermoplastic materials are polystyrene (low cost, lacking the strength and longevity of other materials), ABS or acrylonitrile butadiene styrene (a terpolymer or mixture of compounds used for everything from Lego parts to electronics housings), polyamide (chemically resistant, heat resistant, tough and flexible – used for combs), polypropylene (tough and flexible – used for containers), polyethylene, and polyvinyl chloride or PVC (more common in extrusions as used for pipes, window frames, or as the insulation on wiring where it is rendered flexible by the inclusion of a high proportion of plasticizer). Injection molding machines, also known as presses, hold the molds in which the components are shaped. Presses are rated by tonnage, which expresses the amount of clamping force that the machine can generate. This pressure keeps the mold closed during the injection process. Tonnage can vary from less than 5 tons to 6000 tons, with the higher figures used in comparatively few manufacturing operations.

Molds are typically constructed from hardened steel, pre-hardened steel, aluminum, and/or beryllium-copper alloy. The choice of material to build a mold is primarily one of economics. Steel molds generally cost more to construct, but their longer lifespan will offset the higher initial cost over a higher number of parts made before wearing out. Pre-hardened steel molds are less wear resistant and are used for lower volume requirements or larger components. The steel hardness is typically 38-45 on the Rockwell-C scale. Hardened steel molds are heat treated after machining. These are by far the superior in terms of wear resistance and lifespan. Typical hardness ranges between 50 and 60 Rockwell-C. Aluminum molds can cost substantially less, and when designed and machined with modern computerized equipment, can be economical for molding tens or even hundreds of thousands of parts. Beryllium copper is used in areas of the mold which require fast heat removal or areas that see the most shear heat generated. The molds can be manufactured by either CNC machining or by using Electrical Discharge Machining processes. Molds are built through two main methods: standard machining and EDM machining. Standard Machining, in its conventional form, has historically been the method of building injection molds. With technological development, CNC machining became the predominant means of making more complex molds with more accurate mold details in less time than traditional methods.

II. METHODOLOGY

2.1 Study of the component: The study of the component is the most important to design the mould. It consists of characteristics of component suggested by customer. The important information available is dimensions, number of components required, protection to threads.

2.2 Solid model of component: Solid modelling of component is done using “CATIA V5 R20”.

2.3 Material selection: Proper selection of material to protect the threads. It material having properties as flexible, durable, light in weight & low cost. Material for mould having properties as good machinability, good tensile strength, high melting point, less corrosive, low cost.

2.4 Step by step design Calculations: It is carried out to determine the various design parameters. To determine clamping tonnage, shot capacity, plasticizing capacity & number of cavities.

2.5 Solid modeling of tool: The 3-D modeling of tool is done by using CATIA V5 R20. The required dimensions are based on component design & design calculation.

2.6 Programming: VMC programming is done on NX CAD/CAM. The 2-D program generate with help of CATIA model.

International Journal of Advance Research in Science and Engineering
Volume No.07, Special Issue No.03, February 2018
www.ijarse.com
IJARSE
ISSN: 2319-8354

2.7 Cost Estimation: The cost of mould is based on material quality & machining equipment. Low carbon steel has less cost & good quality. The machining cost is depending on critical design & number of cavities.

III.SOLID MODELING

Core plate: 150x150x50 mm

Cavity Plate: 150x150x50 mm

Ejector top & Ejector bottom Plate: 150x100x15 mm

Bottom plate: 150x150x20 mm

Spacer: 150x70x25 mm

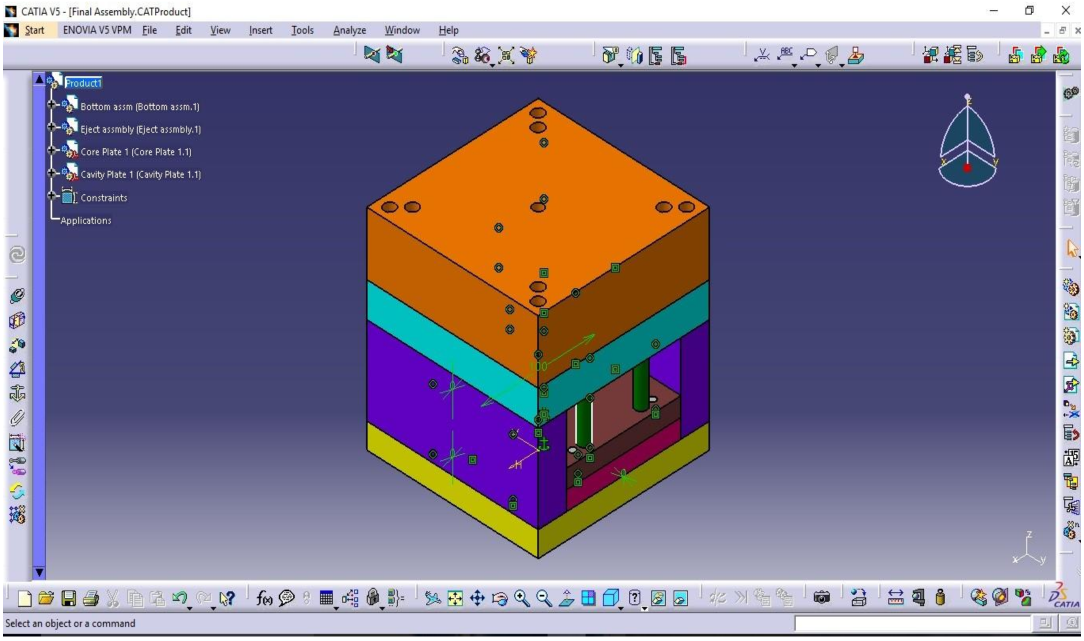

Fig.1. Assembly of mould

IV.DESIGN CALCULATIONS

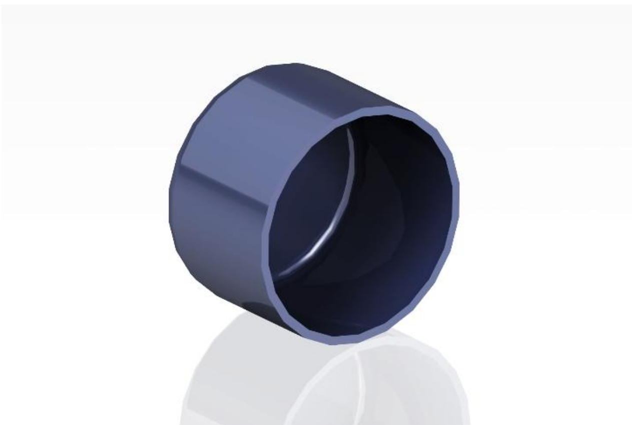

Fig.2. Thread Cover

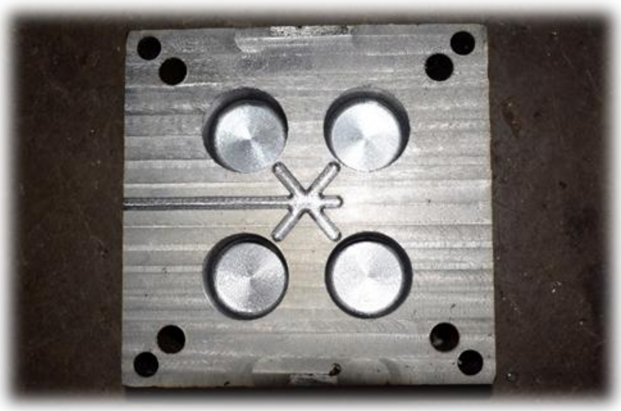

Fig.3. Cavity Plate

4.1 Part Details

Name of the component: Thread Cover

Material: Polypropylene

Shrinkage: 1.5-2%

Volume of component: 4.204 cm^3

Density of material: 0.895-0.905 g/cm^3

Weight of the component: 39.24 g

Number of cavities: 4

Projected area of component: 70 cm^2 (from CATIA V5 model)

4.2 Weight of Moulding

Actual weight of component, (W)

$W = \rho \times V$ ...(1)

W = Actual weight of the component in g

ρ = Density of plastic material = 0.905 g/cm^3

V = Volume of the component, cm3 = 4.204 cm^3 (CATIA V5 model)

W = 0.905 x 4.504

W = 4.08 g

Total weight = W x Number of cavities

Total weight = 4.08 x 4 = 16.32 g

The weight of the sprue and the runner related to the moulding must not generally be neglected. This should be considered in the formula while determining the moulding weight. The moulding weight should be substituted in

the formula and multiplied with the multiplication factor (M.F).

Total weight of single component with feed system = 16.32 x 1.05 = 17.14 g

4.3 Clamping Tonnage

Clamping tonnage required = Total Projected area of the mould × Cavity pressure x no. of cavities ...(2)

Injection pressure required for processing polypropylene to produce an engineering part is 1000 kg/cm^2 (maximum).

1/2 Of injection pressure, as cavity pressure for easy flow materials, 1/3 of injection pressure, as cavity pressure for viscous materials. Polypropylene has good flow-ability, hence 1/2 of the injection pressure, may be assumed as the cavity pressure.

Tonnage required for the component = Total projected area x 1/2 Injection pressure x number of cavities

Tonnage required for the component = 70 × (1/2 × 1000) x 4

Tonnage required for the component = 140000 Kg

Factor of safety of 1.3 (30% of actual tonnage) = 182000 Kg

Minimum machine tonnage required = 182 tonnes = 1813.45 KN

It is suggested that the available machine is of 200T clamping capacity.

4.4 Plasticizing Capacity (p_s)

Plasticizing capacity of the machine is calculated as follows,

Rated plasticizing capacity of the material is:

Plasticizing rate= Plasticizing rate of polystyrene x $\frac{Qps}{Qpp}$

Plasticizing rate of polystyrene = 16.6 g/sec

q_a = Total heat of polystyrene = 57 cal/g

q_b = Total heat of polypropylene = 130 cal/g

$p_s = \frac{16.6 \times 57}{130}$

P_S = 7.28 g/s = 26.21 kg/hr

Plasticizing capacity of the machine polypropylene is 26.21 kg/hr

4.5 Shot Capacity (SC)

The capability of machine is normally expressed in cubic centimeters of swept volume the injection cylinder. The shot is, therefore, the mass of this volume of plastic melt at the plasticizing temperature and pressure. Thus,

Shot capacity (kg) = Swept volume x Density of material x Cont. ...(4)

where,

Constant = correction factor for percent volume expansion of the plastic at the moulding temperature for PP = 0.9 (Crystalline materials).

The screw type is normally rated in terms of swept volume of the injection cylinder = 594 cm3 (from Machine specification).

Density of material = 0.905 g/cm^3

Shot capacity (g) = 594 x 0.905 x 0.9 = 485 g

Shot capacity of the machine is 485 g. Since the shot weight of the component is 17.14 g, the design is safe and production of the component can be carried out without any restrictions.

4.6 Determination of Number of Cavities Based on Shot Capacity

Ns = 0.80 × (Shot capacity/Weight of component) ...(5)

Shot Capacity of the machine for PP is 85 g

Ns= 0.80 x (485/17.14)

Ns = 22.64 = 23 cavities

Depending on shot capacity the mould can be designed to accommodate 23 cavities. But the cost for manufacturing the 23 cavities mould is more. So 4 cavity mould is designed to reduce the cost.

4.7 Determination of Number of Cavities Based on Plasticizing Capacity

$Np = \frac{08 \times Ps \times Ts}{Ws}$

Ps = Rated plasticizing of PP in grams per hour

Ts = Cycle time in seconds

Ws = Weight of the component in grams

Np= (0.80 x 26.21 x 17)/ 17.14

Np = 20.80= 21 cavities

Depending on plasticizing capacity the mould can be designed to 4 cavities. Hence the design is safe.

4.8 Determination of Number of Cavities Based on Clamping Tonnage

Nc $=\frac{C}{Pc \times A}$

Clamping force (C) = 1813.45 KN

Projected area (A) = 70 cm^2

Cavity pressure (Pc) = 45 Mpa

Nc = 5.7 cavities = 6 Cavities

Depending on Clamping tonnage the mould can be designed to 4 cavities. Hence the design is safe.

V. RESULTS & DISCUSSION

From the above design calculations following details are for manufacturing the tool:

5.1 Total weight of single component with feed system is 17.14 g.

5.2 Minimum machine tonnage required is 180 T. It is suggested that the available machine is 180 T Machine.

5.3 Total Weight of material required per hour is 26.21kg/hr and machine plasticizing capacity is 104 kg/hr, hence machine selection is safe.

5.4 Shot capacity of the machine is 485 g. since the shot weight of the component is 17.14 g. the design is safe and production of the Component can be carried out without any restrictions.

5.5 Based on shot capacity, clamping tonnage and plasticizing capacity number of cavities are selected as four.

VI.CONCLUSION

Design of thread cover is used to protect the threads of shaft. Due to this life of threads is increased. For the manufacturing of thread cover injection moulding process is selected. The material of mould is mild steel which easily available at low cost. Also manufacturing cost is less. So low cost injection mould is manufactured. On the basis of design calculations, the number of cavities are selected as Four.

REFERENCES

[1.] Kiran Tom Thomas, and Ramesh Babu “conceptual design of two plate injection mould tool for five pin daimler regulator” International Journal of mechanical engineering & robotics research ISSN 2278 – 0149 Vol. 3, No. 4, October 2014 2014 IJMERR.

[2.] Mr. Prashant A.Dhaware, Prof. M A. Sayeed, Prof. A.M.Kalje, “Design and Analysis of Injection Moulding Die for Churner” IJAIEM Volume 3, Issue 4, April 2014

[3.] Yogendra M. Verma, Shubham B. Kurrewar Mathematical Modeling of Plastic InjectionMould IJSTE - International Journal of Science Technology & Engineering | Volume 2 | Issue 10 | April 2016

[4.] Harshvardhan Patil, Nikhil Deshpande, Sudarshan Bhambore, Kuldeepak Lokhande Design & Development of Plastic Injection Mould for Plastic Clip of School Bag international journal of trend in research and development, Volume 3(3), ISSN: 2394-9333

[5.] Dr. M P Singh, Akhilesh Jaguri Gating System Design Optimization for Injection Moulding, International Journal of Mechanical and Industrial Technology, Vol. 3, Issue 1, pp: (156-163), Month: April 2015 - September 2015

[6.] Jitendra Dilip Ganeshkar, Prof. R.B Patil, Swapnil S. kulkarni design of 'plastic injection mold' for an automotive component through 'flow analysis'(cae) for design enhancement, International Journal of Advanced Engineering Research and Studies E-ISSN2249–8974