International Conference on Innovative Research in Engineering, Management and Sciences
International Journal of Scientific Research in Computer Science, Engineering and Information Technology
© 2019 IJSRCSEIT | Volume 4 | Issue 9 | ISSN : 2456-3307

Design of Injection Moulding Tool for the Component

Shivaprakash S [1], Chetan Kumar D S [2]
[1, 2]Senior Assistant Professor, Department of Mechanical Engineering, New Horizon College of Engineering Bengaluru, India

ABSTRACT

This work deals with the Design and Manufacture of Injection Moulding Tool for the Component Press Button in HIPS material by the Screw type Injection Moulding process. The design and fabrication of a mould for the given component is most challenging task in plastic processing in injection molding. It determines the quality, performance and the profitability of a plastic component. The 3D model of the component is created using solid works software. The various views of different parts involved in mold are taken using this software. These views are converted to 2D drawings for easy dimensioning. Design Protocol is on conventional method, based on tried and tested norms Choice of the most appropriate option for the selection of parting line, feeding system, cooling system, venting, etc. are based on the manufacturing point of view. The above parameters help in development of various parts in mold that are functional, reliable, manufacturability and aesthetically pleasing. Manufacturing involves design of electrodes, process planning, machining and assembly. Complete process planning for each part of the tool is not carried for the sake of brevity. After fabrication of the tool, samples of the component were produced. These samples were inspected to find defects, if any.

Keywords : HIPS, 2D drawings

I. INTRODUCTION

Injection molding is a plastic-forming process used in the production of most (about 70%) of plastic parts. Injection molding is generally used in the high-speed manufacture of low-cost, high- volume products like videocassette cases, plastic cups, or children's toys, etc. The mold defines the shape of the part, as well as the path by which the molten plastic flows from the barrel of the Injection Molding Machine.

The process of injection molding begins with a barrel full of hot, liquid plastic. The plastic is rammed at high pressure into a mold. Once the plastic fills the mold, it is allowed to cool and solidify. The finished part is then extracted (usually automatically) from the mold. No chemical reaction occurs during the molding process. Any reaction that occurs would be a degradation reaction, which should be avoided.

Complex shape of different thermoplastic materials can be formed economically under high production rates. Component obtained by this process will posses high surface finish, low cost and minimum scrap as runners, gates and sprues can be reused.

Most of the tools fabricated involve standardized mold bases, for ease the construction and reduce the manufacturing cost. Only the inserts that give the internal and external shape of the component has to be fitted into the mold base. More care has to be taken while designing and manufacturing the inserts to obtain the required shape and size of the component. Overall tool design should lead to less cost, high productivity and ease of manufacturing.

Press Button is used in adjustment of seat belt in cars. It involves internal projection, which helps to hold seat belt for adjustment. It should posses good surface finish and meet the requirement so as to reduce post-processing operation. Since the material chosen for this component is High Impact Polystyrene (HIPS), Injection molding machine is used for production.

II. SCOPE OF PROJECT

Various approaches have been adopted while designing the tool for producing defect free components.

- Study of various features involved in the component for selection of parting line, gate location, vent locations, cooling circuits, etc.
- Choice of various tooling materials and molding machine to produce large volumes of plastic component.
- Basic calculations for determining cavity wall strength, shot weight, clamping force, cooling and cycle time, etc. are carried.
- 3D modeling of the tool has been done for physical representation before manufacturing.
- Trail and trouble shootings to obtain sound components.

III. OBJECTIVES

The main objective is to develop a protocol of the tool for the given component using scientific approaches. The study is made to design the mold as simple as possible from the cost and manufacturing point of view, simultaneously maintaining the quality of the component within the specified limits.

IV. METHODOLOGY

The basic concept involved in this method is to attain the objective of the systematic and correct tool design; a well-planned approach has been employed. The methodology consists of the following.

- Component analysis.
- Solid modeling of the component.
- Step by step approach to mould design.
- Selection of tooling materials.
- Solid modeling of the tool.
- Tool fabrication, assembly and tryout
- Injection molding defects

TOOL DESIGN

In this section step by step approach to the design of Injection Molding Tool is set based on experience, empiricism and expertise as applied to "Press Button" and various design calculation are given here

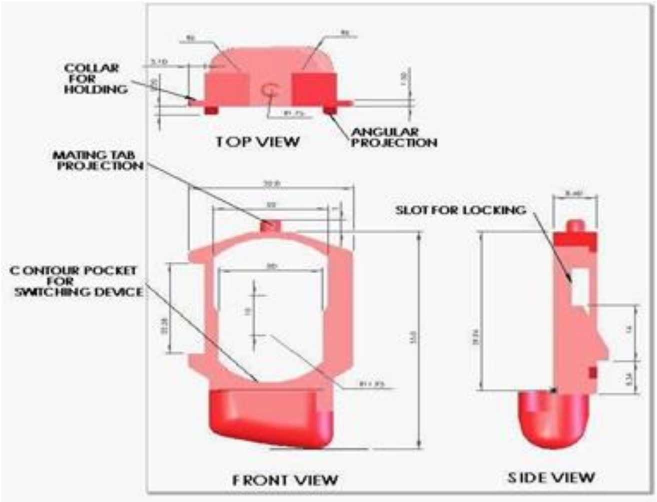
COMPONENT DETAILS

Material : HIPS – (High Impact Polystyrene)
Density : 1.05 gm/cm3
Volume : 4.86 cm3 Mass : 5.00 gms

COMPONENT DESCRIPTION

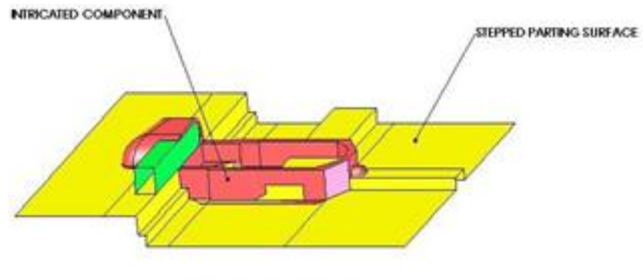
Press button component is used in cars for seat adjustments. The Plastic material used for the press button component is HIPS as per the customer requirement. It involves internal projection, which helps to hold seat firmly while driving the vehicles. It should posses good surface finish and meet the requirement so as to reduce post- processing operation. The Component drawing is received from the customer end. The component has a varying wall thickness with a minimum wall thickness of 1.75mm & maximum wall thickness of 6.36mm. The outer dimensions of the component are 32x55mm & have a total depth of 12.75mm.

The inner surface is the functional surface; therefore there should not be any ejection marks. Isometric view of exterior of the component is shown in Fig.



SELECTION OF PARTING PLANE

The selection of parting plane entirely depends on the shape and geometry of the component. After careful study of the component drawing, a stepped parting plane is chosen. This permits easier ejection of the casting. The criteria for selection of parting plane are detailed are shown in fig



PARTING SURFACE

Weight of the component = 5.0gms

Where, W = Weight of the component (gms).

□ = Density of the plastic material used (gm/cm3) V = Volume of the component (cm3).

b) Moulding weight = Weight of the component X Multiplication Factor = 5.0 X 1.05 = 5.25gms

The weight of sprue & runner related to the moulding generally must not be neglected. This should be considered in the formula when determining the moulding weight. The moulding weight should be substituted in the formula & multiplied with the multiplication factor.

DETERMINATION OF CLAMPING FORCE

Cf = (AP + AR) n X (1/2 to 1/3 of IP) Where, Cf = Clamping force in tonnes.

AP = Projected area of moulding = 57.73X25.50 = 1472.0mm2

= 14.72 cm2

AR = runner area = 15% of projected area of the component

= 0.15 X 14.72 = 2.208 cm2

n = number of cavities = 2

IP = Injection Pressure in N/cm2 Injection Pressure of HIPS = 71 - 215 N/cm2 IP = □1/2 (215) +1/3 (215)□/2 = 89.18 N/cm2

Substituting these values in the above equation

Cf = (14.72 +2.208) 2 X (89.18) Cf = 3019.53 N/cm2

Cf = 3019.53X9.81 = 29621.60 kgs

Cf = 29.62 Tones □ 30 tonnes

DETERMINATION OF NUMBER OF CAVITIES

While deciding the number of cavities, approximate dimensions of the component is taken into account; here approximate dimensions of “Press Component” are 32x55mm. And also shot weight, clamping force, injection pressure and number of components to be produced from the tool, here 100K number to be produced. Horizontal Injection Moulding Machine SP-30T was to be employed for production in the plant.

WEIGHT OF THE COMPONENT

a) Actual weight of the component W = □ X V

W = 1.05 X 4.86

SHOT CAPACITY

Shot capacity of the SP30 machine (screw type) = 44 gms from the machine specification manual.

Actual shot capacity required

Actual shot capacity = {W + (WAR & WAG)} n Where,

W = weight of the component (gms) = 5.25gms

WAR & WAG = Weight of runner & gate area respectively

= 15% of component weight = 0.15 X 5.25

= 0.78 gms 0.80 gms n = number of cavities = 2

Substituting these values in the equation

Actual shot capacity = {5.25 + (0.8)} X 2 = 12.10 gms

As the actual shot capacity is less compared to shot capacity of the machine. Hence SP30T, which is available, can be used.

PLASTICISING CAPACITY

PR of HIPS = PR of Polystyrene X QA/QB

Where, PR = Plasticising rate (gm/sec) PR of Polystyrene = 4.7 gm/sec

QA = total heat content or thermal capacity of Polystyrene

= 57 cal/gm.

QB = total heat content or thermal capacity of HIPS = 58 cal/gm. Substituting these values in the equation

PR of HIPS = 4.7 X 57/58

= 4.61 gm/sec

= 16.59 kg/hr

17.00 kg/hr

The clamping force, shot capacity & Plasticising rate required for the component is well with in the range of SP30T. Therefore SP30T is selected to process the moulding operation.

DESIGN OF CAVITY & CORE INSERT

The minimum wall thickness of Cavity insert is calculated based on the empirical formula given below:

ti = 3 12.P .L4.a

384. E.b.δ

Where, ti = Minimum wall thickness of the insert (mm) P = Maximum Cavity Pressure (N/cm²),

Usually cavity pressure should not operate at above 85% of injection pressure.

Injection pressure for HIPS material = 71 – 215 kg/cm² Therefore, Maximum Cavity Pressure P = 85% X 215

= 181.94 N/cm²

L = Maximum length of component = 57.73mm = 5.773 cm

5.80 cm

a = Maximum cavity depth of component = 8.40 mm = 0.84 cm E = Modulus of elasticity of steel = 2.1 X 106 kg/cm²

= 2140.67X10³ N/cm²

b = Cavity plate thickness = 36mm = 3.6cm

δ = Maximum deflection of side wall = 0.0025 – 0.005 cm Substituting the above values in the equation

ti = 3 12 X 181.94 X (5.80)4 X 0.84

384 X 2140.67 X 103 X 3.6 X 2.5 X 10-3

ti = 0.654 cm ~ 6.54mm

Select the minimum wall thickness of the inserts = 20mm Therefore, by considering the wall thickness & component size, the size of the cavity insert: 66mm X 46mm X 30mm.

Core insert

The length & width of the core insert is taken 66mm X 46mm in order to accommodate the finger cam pins (side core) placed at an angle15 to form the side core for easy ejection, cooling circuit & fasteners. The core inserts thickness taken as 36mm.

DESIGN OF FEED SYSTEM

RUNNER DESIGN

Runner Type: In this design, half round runner is employed to feed the component, because of its ease of manufacturability i.e., Inexpensive, less scrap compared to all other cross-sections.

The runner diameter is calculated by the following formulae:

dr = W X 4Lr

Where, dr = diameter of the runner (mm)

W = weight of the component with losses = 5.25 gms

Lr = length of the runner = 90.0 mm

Substituting these value in the above formulae (4.7)

dr = 5.25 X 490 3.7
= 2.0 mm
The half round diameter of the runner D = 2 X dr D = 2 X 2. = 2.32 mm
Since by considering the Material wastage, Pressure drop & Cooling time. The half round diameter 'D' of the runner is taken as 5mm.

GATE DESIGN

In this design, the Rectangular Edge gate is employed to feed the component.
Since this type of gate is used for two impression rectangular shaped moulding, on two plate moulds and offers easy de-gating & material savings.
i. Gate width: The gate width is calculated by the following formulae
WG = nm X Ac
30

Where, WG = width of the gate (mm). nm = material constant = 0.6 for HIPS
Ac = Total surface area of the component = 3853.57 m WG = 0.6 X3853.57
30
= 1.24 mm ~ 1.00 mm
ii. Gate depth: the gate depth is calculated by the following relation:
h = 0.7nmt Where, h = gate depth (mm) nm = material constant = 0.6 for HIPS
t = average wall thickness of the component = 1.75mm h = 0.7 X 0.6 X 1.75= 0.735 mm ~ 1.00 mm
The depth of the gate controls the time for which the gate remains open. This gate open time must be sufficient for the material to reach the extremities of the impression. Therefore gate depth is taken as 1mm considering de-gating problem, pressure loss.

Therefore gate depth 'h' = 1mm
The land length of the gate should be 1 to 2mm, & taken as 2.00 mm DESIGN OF FINGER CAM PIN
Finger cam pin
Split movement required to clear the component 'Ms'= 5.0mm Finger cam diameter 'Ø' varies from 15 – 25mm. (Taking the average i.e., 20mm as finger cam pin diameter).

Clearance 'C' = 0.5mm/side.
Effective length 'Le' of finger cam can be determined by using the empirical formulae:
Le = Ms + 2C Sin Ø Sin2 Ø
Le = 20 + 2(0.5)

Sin (20) Sin2 (20) Le = 16.17 mm ~ 16 mm
Effective length of the finger cam pin = 16.00mm
Where, Le = effective length of the finger cam pin.

DESIGN OF GUIDE PILLAR AND GUIDE BUSH

The size of the guide pillar is of great importance in the mould. The working diameter of the pillar & the number of guide pillars will depend on the size of the mould & whether or not a side force is likely to be exerted on it. So the guide pillars should withstand side forces to be exerted on it. The moulds with deep & heavy cross- sectional cores exert side thrust & the guide pillars should be strong enough to absorb them without damage.
Guide Pillar
Side thrust Q = ac x hc x Pc
Where, ac = Maximum side of core = 20 mm = 2.0cm
hc = Height of core = 8.5 mm 0.85 cm
Pc = Cavity pressure = 30.58N/cm²
Q = 51.986N

Working diameter (Dp) of guide pillar > 4Q / NPfs
Where, Q = side thrust (Kgs)

NP = Number of pillars = 4
fs = Shear stress = 163.09N/cm² Dp = > 3.18 mm
The size of the mould of the present project is215 X 230 mm, therefore working diameter of the pillar is chosen as 15mm for safe design and to have sufficient strength to withstand side thrust.
Stem diameter of guide pillar = working diameter + wall thickness (say 4 mm)

= 15 + 4 X 2 = 23 mm ~ 25 mm

Collar diameter= Stem diameter of guide pillar + minimum thickness (say 2 mm) = 25 + 2 X 2 = 29 mm ~ 32 mm

Total length of guide pillar = cavity housing thickness + 15mm (guiding length in top plate) + Core housing thickness

= 26 + 15 + 46 + 15 = 102 mm ~ 116 mm

Table for the range of sizes of pillar

Size of the mould (mm x mm)	Working diameter of the pillar (mm)
100 X 100	10
100 X 150	13
150 X 200	16
200 X 250	19
250 X 300	22
300 X 400	24
400 X 600	32
600 X 700	36

Guide Bush

ID of guide bush = Working diameter of guide pillar = 15 mm

OD of guide bush = ID of guide bush + wall thickness = 15 + 2(4) = 23 mm~ 20 mm

Collar diameter= OD of guide bush + minimum thickness (say 2 mm)

= 25 + 2 X 2 = 29 mm~ 25 mm

Length of guide bush = Cavity housing thickness + Top Plate

= 26 + 27 = 53 mm ~ 50 mm Tolerance grade:

Mould plate bore & fitting diameter of the pillar: H7/ k6 Guide bush bore & working diameter of the pillar: H7/g6 DESIGN OF EJECTION SYSTEM

For this component, Pin ejection is chosen for ejecting the component out.

Determination of Ejection Force

Ejection force required to eject the component is determined by using the formula

PE = S x t x Ep x A x / {d (d/2t - d/4t)}

Where, PE = Ejection force required in Kgf

t = Average thickness of component = 0.175 cm

Ep = Elastic modulus of plastic material = 22 X 103 Kg/cm2

A = Total area of contact between moulding and mould faces in line of draw = 38.54 cm2

 = Poisson's ratio of the plastic = 0.4 to 0.5

= Coefficient of friction between plastic and steel = 0.5

d = Circumference of moulding surrounding male core = 136.21 cm S= Thermal contraction of plastic across diameter'd'

= Co-efficient of thermal expansion X temperature difference between softening Point & ejection temperature ºC X d (cm)

= 6.5 X 10-5 X (195 – 20) X 136.21 = 1.549 ~ 2.0

Ejection force required PE = 3.73 kgf ~ 0.0038 tonnes

The ejection force required to eject the component is within in the capacity of SP-30T machine. Therefore design is safe.

DESIGN OF COOLING SYSTEM

In this design work, cooling of cavity insert & core insert are provided. Since the core and cavity inserts are designed in rectangular shape, rectangular cooling circuit is adopted with inlet & outlet. It is the most efficient cooling for this design because as the cooling channel covers the maximum surface area of the component. All drillings within the insert should be interconnected and plugged. The rectangular circuit ensures that the flow-ways are close to all four walls of the cavity, allowing a more even temperature control & highest heat transfer co-efficient will occur when the cooling channels are directly in the cavity or core insert.

a) 1) Heat to be transferred from mold per hour (Qw) Qw = Shot weight X QB X number of shots/hr Where, Shot weight = 5.25gms

QB= total heat content or thermal capacity of HIPS = 58 cal/gm Number of shots/hour = 180(Assume 15 to 20 sec per shot practically possible)

 Qw = 5.25 X 58 X 180 = 54810 cal/hr

2) Heat to be transferred per hour by the cooling system (Qc) In practice, it is assumed that only one-half of the total heat of the moulding is dissipated into mould, which should be taken away by the

Cooling system. The other half of the heat is removed by Conventional modes of heat transfer such as Convection & radiation. Heat to be removed by cooling system = 0.5 Qw

= 0.5 X 54810, Qc = 27405 cal/hr

3) Amount of water to be circulated per hour to dissipate heat (mw)
mw = Qc / {K (Tout - Tin)} Where,

Qc = heat to be removed by cooling system (water) = 27405 cal/hr K = Constant to allow for heat transfer efficiency = 0.64 for direct cooling
Tout = Outgoing water temperature C Tin = Incoming water temperature C Tout - Tin = 3C to 5 C of injection mold will have a significance deviation from the above said values. However, these values can be used as a basis to which the other deviations in the practical designs can be added.
4) Solidifying time (Ts):
The solidifying time is proportional to the square of the wall thickness.
Ts = (QB) t2 / {p (Tmaterial - Tmold) 8} Where,
Ts = Solidifying time (sec)
= Density of the plastic material used =1.05gm/cc
QB = total heat content or thermal capacity of the plastic material = 125 cal/gm.
t = average wall thickness of the component = 0.175 cm
p = Thermal conductivity of plastic material = 7 X 10-4 cal/cm.sec.C
Tmaterial= Injection temperature of material = 270C Tmold = Temperature of mould = 80C
Substituting these values in the above equation (4.18)
Ts= (1.05 125) 0.1752 / {7 10-4 (270 – 80) 8}
Ts = 3.77 sec ~ 4 sec

V. CONCLUSION

Design procedures developed in this work for the production of tool for the component Press Button to be produced through injection molding processes using HIPS material has been successfully validated by conducting tryout operation.
A number of new techniques have been studied. The expertise, experience and empiricisms are followed to greater extent that scientific analysis, logic and hierarchy of decision making developed for this specific task. A tool built which works on initial trial appear to be satisfactory.

The project undertaken has been successfully accomplished by carrying out a design exercise, choosing a steel alloy for the mould with good balance of strength & manufacturability. Engineering the gating to promote soundness. Satisfying technical, economic and delivery schedule requirements. The design of the Injection mould has been approved. The tool is manufactured.

VI. REFERENCES

[1] An Interactive Knowledge-Based CAD System For Mould Design in Injection Moulding Processes by C. K. Mok, K. S. Chin and John K. L. Ho Int J Adv Manuf Technol (2001) 17:27–38 @ 2001 Springer-Verlag London Limited
[2] D Rapid Realization of Initial Design for Plastic Injection Moulds by Maria L.H. Low and K.S. Lee Journal of The Institution of Engineers, Singapore Vol. 44 Issue 4 2004
[3] Decision criteria for computer-aided parting surface design by B Ravi and M N Srinivasan
[4] H.Dominighaus "Plastics for Engineers" Elsevier Applied Science Publishers, 1985.
[5] A.S Athalye, "Injection Moulding", Multi Tech Publishing Co.2004 6.Domonic V Rosoto, Donald V Rosoto, Marlene G Rosoto, "Injection Molding Handbook", kluver academic publishers, 3rd Edition, mw = 27405 / {0.64 (5)} = 8.56 lit/hr 09 lit/hr 2000.
[6] Menges/Mohren, "How to make Injection Molds", 2nd publications,
[7] Whelan, J. P. Goff, "Developments in Injection Molding", Elsevier Applied Science Publishers, Volume 1, 2, 3, 1996.
[8] Irvin And Rubin, "Injection Molding Machines", Hanser publications,.

[9] A guide to injection moulding of plastic by Probodh C. Bolur
[10] Injection mould design fundamentals by a. b. glanvill and e.n.denton, Industrial Press Inc.,
[11] R.G.W. Pye, "Injection Mold Design", Longman Scientific and Technical Publication, 4th Edition, New Delhi,
[12] Gastrow Edited by K. Stoeckhert, "Injection Molds 108 Proven designs", Hanser publications,2 nd Edition,
[13] Technical Directory On Design And Tooling For Plastics", CIPET (Central Institute of Plastics Engineering & Technology), Guindy, Chennai.
[14] "Applications Of Plastics", CIPET, Guindy, Chennai. 16."Data Handbook on Plastics", CIPET, Guindy, Chennai. 17."SPI Plastics Engineering Handbook".
[15] "Tool Engineering Parameters", ISTE (Indian Society of Tool Engineers), Hyderabad.
[16] "Design Manual on Plastic Injection Moulds" Volume – I, Central Tool Room & Training Centre, Calcutta.
[17] Tool engineering parameters from Indian Society of Tool Engineers (ISTE)
[18] S.K Choudhury And A.K Hajra Choudhury,"Elements Of Workshop Technology", Volume 2,Media Promoters And Publishers Pvt Ltd,
[19] H M T, "Production Technology", Tata McGraw-Hill Publishing Company Limited,
[20] ASM HANDBOOK Volume – 4, "Heat Treating".