International Journal of Engineering Research and General Science Volume 4, Issue 3, May-June, 2016
ISSN 2091-2730

Model Based Design of Injection Mould For Charge Air Cooling Inlet Tank

Nilesh D. Khutafale[1], Girish M. Dhote[2],

[1] M.Tech Student, Department of Mechanical Engineering, DBACER , Nagpur

[2]Professor Department of Mechanical Engineering, DBACER, Nagpur

nilesh.khutafale@rediffmail.com, girishdhote@gmail.com

Abstract—

A variety of industrial parts are manufactured by injection moulding using different materials. This research work is focused on design and fabrication of an injection mould for production of charge air cooling inlet tank is used in automobile for reducing NO_x emission to improve fuel economy and power output. Here a Polyamide 66 30% glass filled is used. The moulding tool consists of top plate, cavity plate, core plate, spacer block, ejector plate and the bottom plate. An angular lifting mechanism is also developed to achieve the easy removal of part by maintaining the undercut radius which allows easy flow to media.. The ejector mechanism plays a pivotal role for optimum time allocation for parts ejection from mould. So the ejector mechanism is one of the important criteria in mould design. The scope of this research paper is to investigate the influence on the balance reachable flow length, stresses developed, shrinkage which affects smooth ejection of part from mould.

Keywords: Injection Mould, angular lifting mechanism, ejection of part.Mould flow analysis, Clamping tonnage, shot weight

INTRODUCTION

Plastic is a material that can produce many shapes that can be used by human in routine life. All of plastic products are produce from various type of operation or process. All of product Produces with different type of plastic material depend to needed. Plastics are divided into two distinct groups' thermoplastics and thermo sets.

Plastics can be moulded into various forms and hardened for commercial use. Plastic is perfect for this modern age. It is light, strong, easily moulded and durable. Although plastics are thought of as a modem invention, there have always been "natural polymers" such as amber, tortoise shells and animal horns. These materials behaved very much like today's manufactured plastics and were often used similar to the way manufactured plastics are currently applied.

The plastic product can make from the several processes like injection moulding, blow moulding, compression molding, film insert mounding, gas assist moulding, rotational moulding, structural foam moulding, extrusion and Thermoforming. This work will explain and study more about injection moulding. Injection moulding is a process in which the plastic material is injected into a mould forming a plastic product. Injection moulding is a manufacturing technique for making parts from thermoplastic material. The solid plastic material is fed into an injection moulding machine, heated and then pressed into the mould. In injection moulding, plastic pellets or granules are fed from a hopper into a heating Chamber. A plunger or screw pushes the plastic through the heating chamber, where the material is softened into a fluid state. At the end of this chamber, the resin is forced into a cooled, closed mould. Once the plastic cools to a solid state, the mould opens and the finished part is ejected. Standard Two Plate Mold Injection moulding is very widely used for manufacturing a variety of parts, from the smallest component to entire body panel. It is the most common method of production, with some commonly made items including bottle caps and outdoor furniture. Plastic moulding products can be seen everywhere such as plastic tubes, grips, toys, bottles, cases, accessories, kitchen utensils and a lot more. The mould is made by a mould maker from tool steel, usually either steel or aluminum, and precision machined to form the features of the desired part. Mould is used to produce desire product that we needed. Many elements are involved in mould such as feeding, cooling and injector system. In modern technology, CAD software can be used to design mould and after that perfiune machining raw material to produce complete mould. Feeding system is important element for plastic flow in injection mould.The component inlet tank is made of (PA66 GF-30) polyamide 30 % glass filled have high mechanical strength and superior resistance to wear and organic chemicals.

Charge air cooling is an important technology for reducing N0x emission which also has a number of other benefits. Fuel economy, power output and the maximum injection rates an engine can sustain are improved through charge air cooling. 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. 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. Design should be in such a way that it must also accommodate any complex features on the part, such as undercuts or threads, which will require additional mold pieces. Most of these devices slide into the part cavity through the side of the mold, and are therefore known as slides, or side-actions. The most common type of side-action is a side-core which enables an external undercut to be molded.

International Journal of Engineering Research and General Science Volume 4, Issue 3, May-June, 2016
ISSN 2091-2730

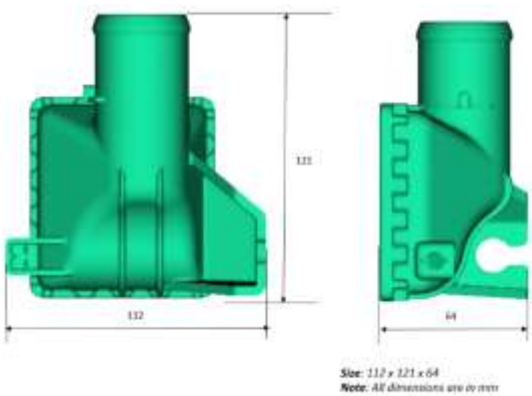

Fig. 1. Component Details

Fig .2 Angular lifter

Fig. 3 Mould view

DETAILS OF INJECTION MOULDING TOOL:

MOULD OR DIE ARE THE COMMON TERMS USED TO DESCRIBE THE TOOLING USED TO PRODUCE PLASTIC PARTS IN MOULDING. THE VARIOUS TYPES OF INJECTION MOULDSARE TWO PLATE MOULD, THREE PLATE MOULD, SPLIT CAVITY MOULD, STRIPPER PLATE MOULD AND HOT RUNNER MOULD. HERE FOR THESE COMPONENT TWO PLATE MOULDS IS SELECTED BY CONSIDERING COST AND ECONOMICAL FACTORS. THE DIFFERENT PARTS OF INJECTION MOULDING TOOL WITH MATERIALS IS LISTEDIN TABLE 1.

Sr.No.	MouldElement	Material
1	CAVITY PLATE	1.2738(special P20)
2	CORE PLATE	1.2738(special P20)
3	CORE BACK PLATE	EN31
4	EJECTOR PLATE	EN31
5	EJECTOR BACK PLATE	EN31
6	SPACER BLOCKS	EN31
7	TOP PLATE	EN31
8	BOTTOM PLATE	EN31
9	GUIDE PILLAR	EN31
10	GUIDE BUSH	EN31
11	SPRUE BUSH	EN31
12	LOCATOR RING	EN31

Table No. 1

DESIGN OF INJECTION MOULDING TOOL FOR **CAC INLET TANK :**

2.1. Modelling and design mouldof CAC inlet tank. It is decided to design and Simulate Injection moulding tool with ejectors assembly, baffle cooling channel and angular lifting mechanism..Based upon the model of CAC inlet tank, the different parts of the injection moulding tool is identified and a model of injection moulding tool is created in Creo parametric 2. For design of injection moulding tool for CAC inlet tank, the following steps are used.

Part details :

Name of the component : CAC Inlet Tank
Material: Material : PA 66 with 30% glass fibers (Durethan AKV 30 H2.0 901510)
Shrinkage 0.2- 1.1%
Volume of the component 71.48 cm3 (From CAD model)
Density of the material : 1.39 g/cm3
Projected area of component: 114.12 cm2 (From CAD model)
Weight of the component (W)= Volume X Density

W= V X p= 71.48 X 1.383 = 98.931 gm.

Total weight = W X Number of Cavities = 98.931 X 01 = 98.931 gm.
Shot Weight = Total weight of the component + Total weight of the feed system

Total weight of the feed system = 10% of component weight= 0.1 X 98.931= 9.893 gm.
Shot Weight = 98.931+ 9.893= 108.824 gm.
Clamping Tonnage :
Clamping tonnage required = Total projected area of the mould X Cavity pressure X no. of cavities
Injection pressure required for processing polyamide 66 with 30% glass filled to produce an engineering part is 1000 gm/ cm^3 maximum.
½ of injection pressure, as cavity pressure for easy flow materials as polyamide 66 with 30% glass filled has good flow ability hence ½ of injection pressure may be assumed as the cavity pressure.
Clamping tonnage required = Total projected area of the mould X Cavity pressure X no. of cavities
= 114.12 X (1/2 X 1000) X 1 = 57060 kg
Factor of safety of 1.3 (30% of actual tonnage) = 57060 X 1.3 = 74178 kg.
Minimum machine tonnage required = 74. 178 tonne.

Plasticizing Capacity (p_s)
Plasticizing capacity of the machine is calculated as follows:
Rated Plasticizing capacity of the material is:

Plasticizing rate = with material PS X $\frac{q_a}{q_b}$

Plasticizing rate of polystyrene = PS = 16.6 g/sec
q_a= 57 cal/g (total heat of polystyrene)
q_b= 135 cal/g (total heat of polyamide66)
P_s = 16.6 X 57 / 135
P_s= 7.00 g/s = 25.2 kg/hr
Plasticizing capacity of the machine for polyamide 66 with 30% glass filled is 25.2 kg/hr.
Machine available with capacity of 52 g/s = 187.2 kg/hr.

Shrinkage allowance: Shrinkage is inherent in the injection moulding process. Shrinkage occurs because the density of polymer varies from the processing temperature to the ambient temperature. The shrinkage factor will depends on Plastic material, Processing condition, Product design, Mould design. Shrinkage allowance of Nylon 66 is 0.5% considered. The dimensions of the cam bush has been modified with considering shrinkage allowance as shown in Table 2. This total dimensions are incorporated in the mould designing of core and cavity plate.

Sr.No.	Actual Dimension in mm	0.5% Shrinkage allowance in mm	Total Dimension in mm
1	78.7	0.393	70.093
2	85.5	0.4275	85.927
3	36	0.18	36.18
4	35	0.175	35.175
5	40	0.2	40.20
6	42	0.21	42.21
7	89.4	0.447	89.847
8	15	0.075	15.075
9	49	0.245	49.245
10	22	0.11	22.11
11	68	0.34	68.34
12	23.7	0.1185	23.8185

Table No. 2

Calculation for wall thickness of core /cavity Inserts :
Insert wall thickness, δ,

$$\delta = \sqrt[3]{\frac{CPd^4}{Ey}} \text{ mm}$$

c = constant based on ratio of cavity length to depth = 0.111

P =maxi.Cavity pressure = 630kg/cm^2
d= Depth of cavity wall = 6.4cm
E = Modulus of elasticity = 2.1× 10^{6}kg/cm^2
Y = Permissible deflection for the insert = 0.005cm

$$\delta = \sqrt[3]{\frac{0.111 X 630 X 6.4^4}{2.1 X 10^6 X 0.005}} = 2.235\text{cm} = 22.35 \text{ mm}$$

Design of guide pillar :

Guiding diameter of the guide pillar, d_p

$$d_p = \sqrt{\frac{4XQ}{\pi X N_p X f_s}}$$

Q = side thrust

N_p = Number of guide pillars = 4 numbers
f_s=Working shear stress for the guide pillar material, kg/mm^2
Side thrust,
Q = d_i X h X P_c
d_i=Height of the core, mm
h = Maximum side of the core, mm
P_c = Pressure in the cavity, kg/cm^2
Q = [6.4 X 12.1 X 630] = 48787.20kg
Substituting the value of the side thrust induced, we get the minimum diameter of the guide pillar,d,

$$d_p = \sqrt{\frac{4X48787.20}{\pi X 4 X 1600}} = 3.11\text{cm} = 31.1\text{mm}$$

Feeding system Design:

Runner design :

The runner diameter is calculated by the following formulae

$D = \sqrt{W} X \sqrt[4]{L} / 3.7 -$ Where, W = weight of the component =108.82gm

L = Length of the runner =121 mm
Substituting the values in equation
Diameter of the runner.

$$D = \sqrt{98.931} X \sqrt[4]{121} / 3.7$$

D= 8.91 mm
Therefore diameter of the runner =8.91mm

Gate Design :-

According to the size and shape in the design, Edge gate is employed to feed the component.
To find gate width

$$W = \frac{nX\sqrt{A}}{30}$$

Where, n = constant = 0.9 for polyamide 66 with 30 % glass filled material.
A= Total surface area of the cavity = 14800mm^2

$$W = \frac{0.9X\sqrt{14800}}{30} = 3.64\text{mm}$$

To find Gate depth
Gate depth, h_g = Avg width of gate/Avg thickness of component = 3.64/2.5=1.45mm

Mould cooling calculations:

Calculations are done based on coolant required and heat transfer rate, as follows.

Heat to be transferred from mould per hour (Q):

Q = n X m X qb
Where,
Q = Heat to be transferred per hour (cal/hr)
m =Mass of the plastic material injected into the mould per shot (g) = 108.824 g
n = number of shots per hour (72shots/hr)
qb = Heat content of plastic material,
for polyamide 66 with 30% glass filled =130 cal/g
Q = 60X 108.824X 130
Q = 1018.555KCal /hr
But in practice heat is removed by three ways Conduction, Radiation, Convection.
It is found in practice, that approximately 50 % of the total heat input is carried away by the water cooling systems in moulds.
Therefore amount of heat removed by cooling water is
Qd = 0.5 X Q = 0.5 X 1018.555= 509.277 K Cal/hr

Amount of water to be circulated per hour to dissipate heat (mw) :
Amount of water to be circulated to remove 50% of Heat is calculated as,

$$m_w = \frac{QX0.5XS}{K(T_{out}-T_{in})} = \frac{1018555.2X0.5X4.186}{0.64X5} = 666.198 \text{ Kg/hr}$$

m_w= 666.198 Lits/hr = 11.1033 lits/min
Where, K = Thermal conductivity of water
K = 0.64 for Cooling channels bored in cavity plate or male core
K = 0.5 for Cooling channels bored in back plate
K = 0.1 for Cooling channels in copper pipe
Tout = Outgoing water temperature ^{0}C
Tin = Incoming water temperature ^{o}C
S_W =Specific heat of water=4.186 J/gm^{o}C
m_w =Amount of water required to remove 50% of heat.
Assuming a reasonable temperature difference of Tout-Tin = 5 ^{0}C for water

CONCLUSION:

The work, deals with the designing of injection mould tool of a CAC inlet tank. The designing was carried out with Creo parametric 2. Throughout the project, an attempt has been made to understand the variables associated with the design of an injection mould. The designing process started with a basic tool layout. Core and cavity extraction helped in understanding of the manufacturing process Themould designed, has made it possible to produce high quality product at minimum cost. Defects can be minimized through improved design of the mold with the study of simulation of flow through the mold. Product was produced with less number of defects and according to specifications mentioned, keeping in notice the economy factor.

REFERENCES:

[1] M. Stanek, D. Manas, M. Manas and O. Suba "Optimization of Injection Molding Process" International Journal Of Mathematics And Computers In Simulation Issue 5, Volume 5, 2011, PP413-421.
[2] AlirezaAkbarzadeh and Mohammad Sadeghi "Parameter Study in Plastic
Injection Molding Process using Statistical Methods and IWO Algorithm" International Journal of Modeling and Optimization, Vol. 1, No. 2, June 2011.
[3] Mohd.rizwanHamsin, AzuddinMamat and Aznijar Ahmad-Yazid" Design and analysis Of Multi-Cavity Traditional and H-Branching Runners For plastic injection Mold" International journal of Design and Manufacture.
[4] S. H. Park, W. I. Lee, S. N. Moon, Y.E. Yoo, Y. H. Cho "Injection molding micro patterns with high aspect ratio using a polymeric flexible stamper" EXPRESS Polymer Letters Vol.5, No.11 (2011) 950–958 DOI: 10.3144/expresspolymlett.2011.93.
[5] Wei Guo,*Lin Hua Huajie Mao and ZhenghuaMeng" Prediction of warpage in plastic injection molding based on design of experiments "Journal of Mechanical Science and Technology 26 (4) (2012) 1133~1139 DOI10.1007 /s12206-012-0214-0.
[6] R. Mani, R. Surace, P. Ferreira, J. Segal, I. Fassi and S. Ratchev**"**Process Parameter Effects on Dimensional Accuracy of Micro-Injection Moulded Part" J. Micro Nano-Manuf. 1(3), 031003 (Aug 12, 2013) (8 pages) doi:10.1115/1.4025073.

[7] Wen-Chin Chen Gong-Loung Fu Pei-HaoTaiWei-Jaw Deng" Process parameter optimization for MIMO plastic injection molding via soft computing "Graduate School of Industrial Engineering and System Management, Chung Hua University No. 707, Sec. 2, Wu Fu Rd., Hsinchu 30012, Taiwan Graduate Institute of Technology Management, Chung Hua UniversityNo.707,Sec.2,WuFuRd.,Hsinchu30012,Taiwan.

[8] P.K. Bharti "Recent methods for optimization of plastic injection molding process a retrospective and literature review" International Journal of Engineering Science and Technology Vol. 2(9), 2010, 4540-4554.

[9] Gantar, G. – Glojek, A. – Mori, M. – Nardin, B. – Sekavčnik, M.GašperGantar - Andrej GlojekMitja Mori," Resource Efficient Injection Moulding with Low Environmental Impacts" Strojniškivestnik - Journal of Mechanical Engineering 59(2013)3, 193-200 DOI:10.5545/sv-jme.2012.661.

[10] T. Stachowiak*, T. Jaruga[10] "Structure of gas-assisted injection moulded parts" Department of Polymer Processing and Production Management, Czestochowa University of Technology, Al. ArmiiKrajowej 19c, 42-200 Częstochowa, Poland.

[11] SharifahImihezri Syed Shaharuddin, Mohd. SapuanSalit, Edi SyamsZainudin "A Review of The Eect of Moulding Parameters on the performance of Polymeric Composite Injection Moulding", Turkish J. Eng. Env. Sci.30 (2006), University Putra Malaysia, Department of Mechanical and Manufacturing Engineering 43400 Serdang, Selangor-MALAYSIA.

[12] Technical Directory on design and tooling for plastics, CIPET.