Scientific Journal Impact Factor (SJIF) : 1.711
International Journal of Modern Trends in Engineering and Research
IJMTER
ISSN (ONLINE) : 2349 - 9745
ISSN (PRINT) : 2393 - 8161

DESIGN OF PLASTIC INJECTION MOULD TOOL FOR AIR FILTER BOX BOTTOM COVER

Madhukumar K[1], Sampathkumar L[2], Nataraj M N[3], Kumarswamy R[4]

[1,2,3,4] *Department of Mechanical Egineering, Sir M. Visvesvaraya Institute of Technology, Bengaluru-562157*

Abstract— Injection molding is one of the growing molding process in the plastic field. It is a cyclic process of forming plastic into a desired shape by forcing the material under pressure into a cavity. The shaping is achieved by cooling (thermoplastics) or by a chemical reaction (thermosets). It is one of the most common and versatile operations for mass production of complex plastics parts with excellent dimensional tolerance. It requires minimal or no finishing or assembly operations. In addition to thermoplastics and thermosets, the process is being extended to materials such as fibers, ceramics, and powdered metals, with polymers as binders. Molds basically consist of core and cavity plate, core and cavity back plate, ejector plate, stripper plate, sprue, gates etc. Injection molding is preferred where tight tolerance, good quality and high rate productions are required. This paper presents the design of a plastic injection mould for Air filter box bottom cover. The technique, theory, methods as well as consideration needed in designing of plastic injection mould are presented. Design of mould is carried out using wildfire Pro-E 3.0. Poly Propylene talc filled [PPTF] was used for production of this component. The tool was designed to produce a good quality component considering the ease of manufacturability, assembly and Positive ejection of the component by optimizing time and cost. Component requires two side cores. Hydraulic actuation was used in the tool construction. Tool design and development is a specialized and critical area. The tool design should match the machine specification and should be accurate and economical for successful life of a component or product.

Keywords- Product design, Model study, Tool design and assembly, Side cores, Injection molding.

I. INTRODUCTION

Plastic industry is one of the world's fastest growing industries, ranked as one among few billion-dollar industries. Almost every product that is used in daily life involves the usage of plastic and most of these products can be produced by plastic injection molding method. Plastic injection molding process is well known manufacturing process to create products with various complex shapes and geometry at low cost .The plastic injection molding process is a cyclic process with four significant stages. These stages are filling, packing, cooling and ejection. The plastic injection molding process begins with resin feeding and appropriate additives from the hopper to heating/injection system of the plastic injection molding machine. This is the "filling stage" in which the mould cavity is filled with hot melted polymer at injection temperature. After the cavity is filled, in the "packing stage", additional polymer melt is packed into the cavity at a higher pressure to compensate the expected shrinkage as the polymer solidifies. This is followed by "cooling stage" where the mould is cooled until the part is sufficiently rigid to be ejected. The last step is the "ejection stage" [2]. In which the mould is opened and the part is ejected, after which the mould is closed again to begin the next cycle.

The design and manufacture of injection molded polymeric parts with desired properties is a costly process dominated by empiricism, including the repeated modification of actual tooling. Among the task of mould design, designing the mould specific supplementary geometry, usually on the core side, is quite complicated by the inclusion of projection and depression [3].

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 [3]. 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.

1.1 Injection moulding cycle

The sequence of events during the injection mould of a plastic part is called the injection moulding cycle. The cycle begins when the mould closes, followed by the injection of the polymer into the mold cavity. Once the cavity is filled, a holding pressure is maintained to compensate for material shrinkage. In the next step, the screw turns, feeding the next shot to the front screw. This causes the screw to retract as the next shot is prepared. Once the part is sufficiently cool, the mould opens and the part is ejected [1]. Typical cycle times range from 10 to 100 seconds and are controlled by the cooling time of the thermoplastic or curing time of the thermosetting plastic material.

Injection moulding is a cyclic operation shown in fig .1. The cycle consist of

- Mould close and clamp, (few seconds -depends on machine speeds).
- Injection - Fill (speed) phase, (few seconds)
- Switchover and Pack (pressure) phase,(few seconds)
- Cooling time, (40 to 60% of cycle time)

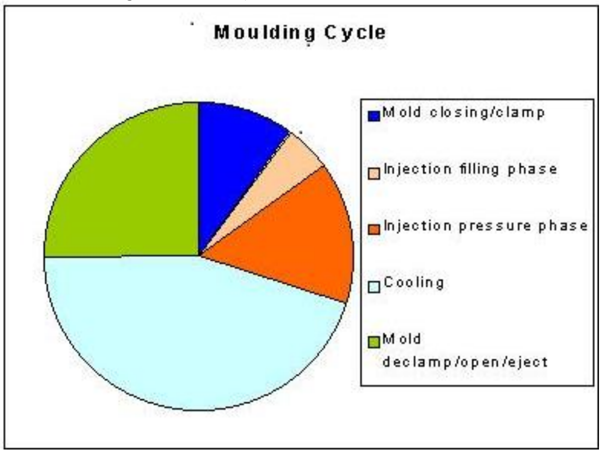

Figure 1. Injection moulding cycle

II. OBJECTIVES OF THE STUDY

The main objective of the study is to design the Injection Mould tool to produce good quality component economically. Also;

1. Model study including, detect and fix problematic zones.
2. The study of selected materials has been done, to know its physical and mechanical properties associated with moulding material and moulding characteristics that influence tool design.
3. Apply a shrinkage that corresponds to the part material, geometry and moulding conditions.
4. Make conceptual design of mould.
5. Design calculations
6. Generating the mould assembly with standard components such as mould base, ejector pins, sprue bush, screws, fittings and other components creating corresponding clearance holes.

7. Modification of Mould according requirement.

III. MODEL STUDY AND MODELLING OF COMPONENT

Model study includes identifying the criticality in component, following are the criticality involved in component

- Require two side cores. (shown in figure.2 marked in red line),
- Proper ejection method required to eject the component because using of large side core.
- Bosses at the sides needs to proper placement of screw inserts.

Component is modeled using the software wildfire pro-e 3.0. Component has a rectangular structure with dimensions: 240mm (length), 160mm (width), 241 mm (height), Thickness of wall is 2.5 mm. Also component got one round opening, sleeves, bosses and ribs at sides shown in the figure 2. Other details of model are given below

Component name: Air filter box bottom cover
Component material: PPTF (polypropylene with talc filled)
Shrinkage : 1.0-1.1%
Max. Wall thickness of the component: 2.5 mm
Moulding type: single Cavity injection mould tool
Specific gravity : 1.20 g/cm^3
Projected area of component: 38.4 cm^2 (From CAD model).

The injection temperature, time and pressure were 210^O C, 12.41Sec and 180 MPa, respectively are obtained by simulation technique.

PPTF(Poly Propylene talc filled)-20%Hypolene 8021M is used for the production of this component. Hipolyene 8021 M is mineral modified PP compound intended for injection moulding of auto interior. Talc is added to poly propylene to achieve Chemical & Heat Resistance, Impact absorbing strength, Dimensional stability, Stiffness, hardness, tensile strength, Electrical insulation properties. Fig. 2 shows the 3D model of air filter box bottom cover component.

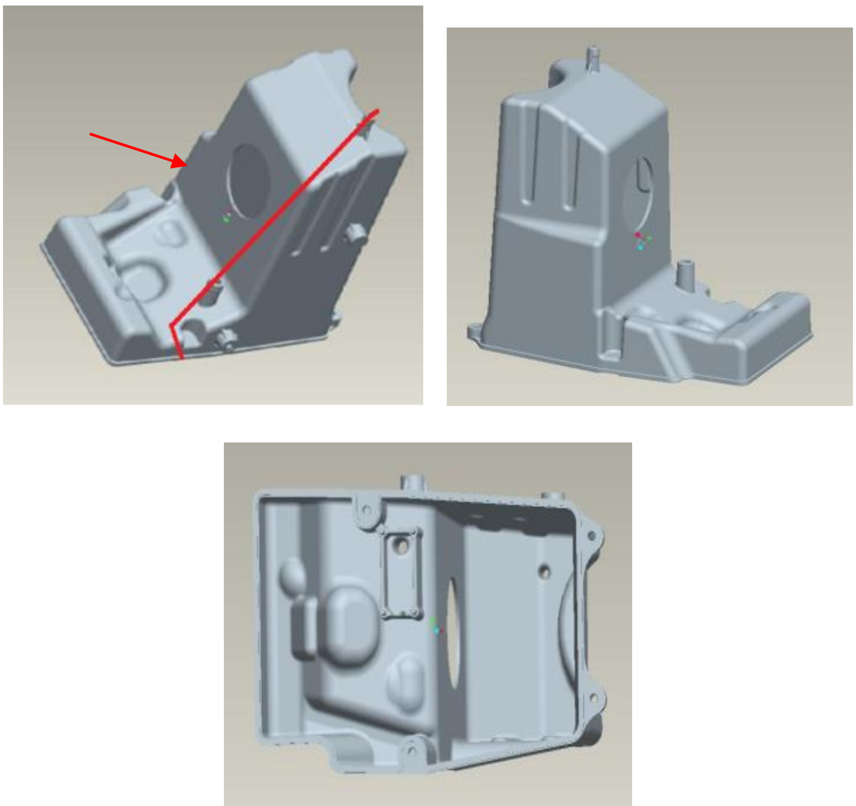

Figure 2. Air filter box bottom cover 3D model

IV. DESIGN OF MOULD

This section describes the design aspects and other considerations involved in designing the mould to produce air filter box bottom cover. The material used for producing the plastic injection mould for air filter box bottom cover was P20 (grade is HH-1.2738) for inserts and C45 for mould base.

Three design concepts had been considered in designing of the mould including:

i. Three-plate mould (Concept 1) having two parting line with single cavity. Not applicable due to high cost and complicated profile of component.

ii. Two-plate mould (Concept 2) having one parting line with double cavities with gating and ejection system. Not applicable because not possible to place the side cores and its actuation system.

iii. Two-plate mould (Concept 3) having one parting line with single cavity without gating system. Eliminates degating problems and provides comfortable space for side core actuation.

While designing of the mould third concept had been applied. Initially, the mould was designed based on the platen dimension of the plastic injection machine used. There was a limitation of the machine, which is the maximum area of machine platen and is given by the distance between two tie bars. The distance between tie bars of the machine is 530 mm. Therefore, the maximum width of the mould plate should not exceed this distance.

Basically, core and cavity extraction was done on the basis of the criticality of the component, after core-cavity extraction from specified modeling software, mold base also modeled on the same software, finally core cavity inserts are assembled into the mold base. Cooling circuit, ejection system, lifting arrangements are designed as per component requirements.

4.1. Factors to be considered during designing of injection moulding tool

- Design and material of components
- Number of components required
- Selection of Injection moulding machine
- Number of cavities
- Type of tool
- Selection of parting line
- Positioning of core and cavity
- Ejection system
- Designing of layout
- Fool proofing arrangements
- Cooling elements
- Tool life
- Tool cost

4.2. Design calculation [2]

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, plastisizing 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.

4.2.1. Weight of Component

Thearitical:

Actual weight of the component, (W)

$W = \rho \times V$

Where, W =Actual weight of the component in grams,

ρ =Density of plastic material, = 0.946 g / cm^3.

V= Volume of the component = 356.79 cm^3 (from CAD model).

$$W = 0.946 \times 356.798$$

$$\mathbf{W = 337.53 \approx 338\ g.}$$

The weight of the sprue related to the moulding must not be neglected. This should be considered in the formula while determining the moulding weight.
Weight of sprue =3.7680 g (from analysis)
Total weight of the moulding = weight of component + sprue weight
= 337.53+3.7680 ≈ **341.298 g**
Values from analysis:

Total volume	=	356.7900 cm^3
Volume filled initially (sprue)	=	2.3864 cm^3
Volume to be filled	=	354.4040 cm^3
Total weight	=	353.7680 g
Weight of sprue	=	3.7680 g

4.2.2 Clamping tonnage

Clamping tonnage required = Total Projected area of the mould × Cavity pressure
Total Projected area of the mould (CAD model) = 384cm^2
Injection pressure required for processing polypropylene to produce an engineering part is 1836 kg / cm^2.

$^1\!/_2$ Of injection pressure, as cavity pressure for easy flow materials,

$^1\!/_3$ of injection pressure, as cavity pressure for viscous materials.

PP has good flow-ability, hence $^1\!/_2$ of the injection pressure, may be assumed as the cavity pressure.

Clamping Tonnage = Total projected area x No. of cavities x $^1\!/_2$ of Inj. pressure

= 384 x 1 x (1/2 × 1836)
Tonnage required for the component = **352,512 Kg**
Minimum machine tonnage required = **350 T**
Hence the machine used is 350 Tonnage capacity

4.2.3. Plasticizing capacity

Plasticizing capacity of the machine is calculated as follows,
Rated plasticizing capacity of the material is;

Plastisizing rate of PP = Plastisizing rate of Ps$\times \frac{Q_{PS}}{QS}$

Plasticizing rate of polystyrene, = 16.6 gm/sec =59.76 Kg/hr (From Machine Specification)
q_a = Total heat of polystyrene =57 cal/g
q_b = Total heat of PP = 132 cal/g

$$P_P = \frac{16.6 \ X \ 57}{132}$$

P_P = 7.16 g/s = 26 kg/hr
Plasticizing capacity of the machine for PP is 50 g/sec.

4.2.4. Shot capacity

(type of machine selected is screw type)

Shot Capacity	= Swept Volume x ρ x Constant
Swept Volume	= 540 cm^3
ρ	= Density of the plastic material

Constant = 0.92 (for amorphous); 0.82 (for crystalline)

Shot Capacity = 540 x 0.946 x 0.82
= 418.8 g

4.2.5. Determination of number of cavities
According to component criticality and shape, single cavity mold is preferred.

4.2.6. Calculation for wall thickness of Core / Cavity inserts
Criterion - Strength of materials;
Insert wall thickness, δ,

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

C = constant based on ratio of core length to depth, = 0.142
P = Cavity pressure = 918 kg/cm^2.
d = Depth of core wall = 9 cm.
E = Modulus of elasticity, = 2.1 × 10^6 kg / cm^2.
y = Permissible deflection for the insert, = 0.005 cm.

$$\delta = \sqrt[3]{\frac{0.142 X 918 X 9^4}{2.1 X 10^6 X 0.005}} \text{ cm}$$

δ = 4.33 cm = 43.3 mm

Table 1.2. Value of C for different L/D ratio

Ratio of the length of Cavity wall to the depth of Cavity wall (L/d)	Values of 'C'
1	0.044
1.5	0.084
2	0.111
3	0.134
4	0.140
5	0.142

4.2.7. Design of Guide Pillars

Guiding diameter of the guide pillar, d_p

$$d_p = \sqrt{\frac{4 \times Q}{\pi \times N_p \times f_s}}$$

Q = side thrust
N_p = Number of guide pillars.
f_s = Working shear stress for the guide pillar material, kg / mm^2
Q = $d_i \times h \times P_c$
d_i =Height of the core =24.1cm
h =Maximum height of the core =29.1cm
P_c =Pressure in the cavity =918 kg/cm^2
Q = (24.1X 29.1 X 918)

Q = 643803 Kg

Substituting the value of the side thrust induced, we get the minimum diameter of the guide pillar, d_p,

d_p = 11.31 cm = 113 mm

A standard diameter of the guide pillar is assumed to be **113 mm.**

4.2.8. Hydraulic cylinder calculation

Hydraulic cylinders are required for side cores actuation, and some important parameters are calculated as follows,

4.2.8.1. Cylinder one (large side core)

Core under cut =35mm
Fafety value added =20mm
Movement of side core required =35+20=55mm
Bore size =Ø40mm
Rod size =Ø20mm
Working pressure=250bar =25N/mm²

$$P = \frac{F}{A}$$

Where, P = Working pressure of cylinder,N/mm²
F = Force required for the movement of core,N
A= Area of the core,mm²

Area (A) =140 x 254.157=35581.98mm²

Force ,F = P x A = 25x35581.98 = 889549.5N

Force required for core movement is = **889549.5N**

4.2.8.2 Cylinder one(round side core)

Core under cut =2.5mm
Safety value added=15mm
Movement of side core required=2.5+15=17.5mm ≈ 20mm
Bore size =Ø40mm
Rod size =Ø20mm
Working pressure =160bar =16N/mm²

$$P= \frac{F}{A}$$

Where, P=Working pressure of cylinder,N/mm²
F=Force required for the movement of core,N
A=Area of the core,mm²

$$\text{Area },A= \frac{\pi d^2}{4}$$

Where,d=diameter of core=20mm

A=314.2mm²

Force, F=PxA=16x314.2=5027.2N

Force required for core movement is = **5027.2N**

4.2.9 Mould cooling calculations

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

4.2.9.1. Heat to be transferred from mould per hour (Q)

$Q = n \times m \times q_b$
Where, Q = Heat to be transferred per hour (cal/hr)
m =Mass of the plastic material injected into the mould per shot (g) = 360 g
n = number of shots per hour (60shots/hr)
q_b = Heat content of plastic material, for =132 cal/g
Q = 60X 360X 132
Q = 2851.2KCal /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
Q_d = 0.5 X Q = 0.5 X 2851.2 = **1425.6 K Cal/hr**

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

$$m_w = \frac{Q \times 0.5S}{k(T_{out} - T_{in})}$$

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
T_{out} = Outgoing water temperature ^oC
T_{in} = Incoming water temperature ^oC
S_W =Specific heat of water=4.186 J/gm ^oC
m_w =Amount of water required to remove 50% of heat.
Assuming a reasonable temperature difference of $T_{out}-T_{in} = 5^o C$ for water

$m_w = \frac{2851200 X .5 X 4.186}{0.64 X (5)}$ = 482510.7692 g/ hr = **482.51 kg/ hr** = 651.38 litres/ hr
= **10.85 litres/ minute.**
Calculations are used during the mould design,for proper incorporation of dimensions and data required for tool design.

V. TOOL ASSEMBLY
Tool assembly is done in modeling software,includes the fixing of extracted core and cavity inserts into the mould base, after assembly 3D models are converted into the 2D drawings for manufacturing process.

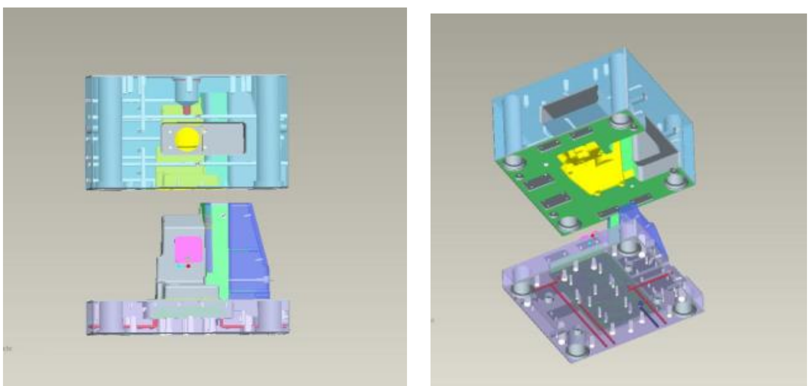

Figure 3. Extracted Core–cavity (exploded view)

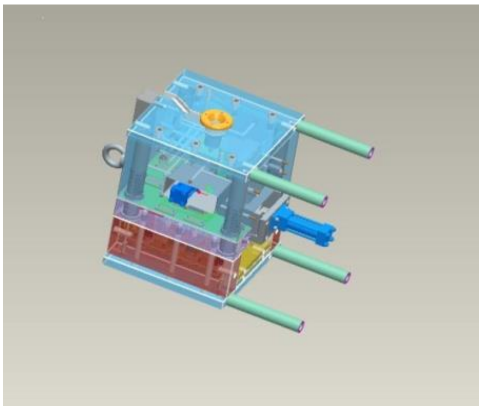

Figure 4. 3D solid modelling of mould assembly

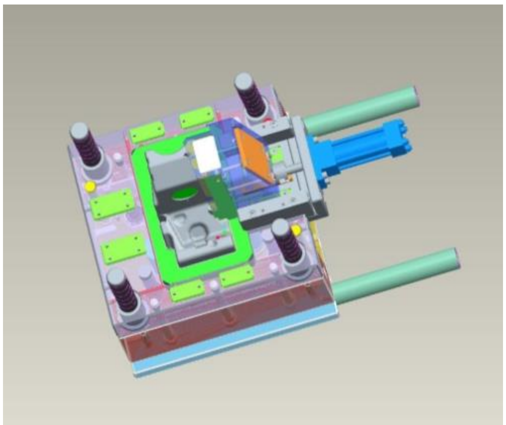

Figure 5. Moving half

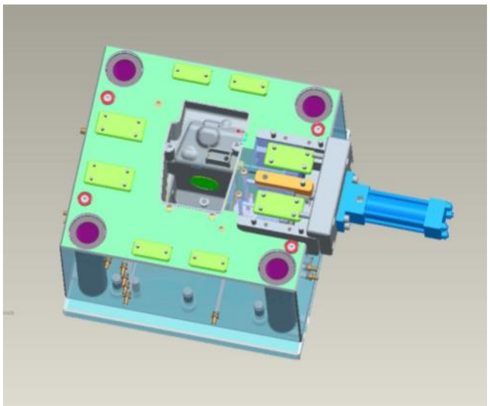

Figure 6. Fixed half

VI. SIDE CORE

Side core is a local core which is normally actuated by using hydraulic cylinders or finger cam. In this design hydraulic cylinders are used, cylinders front flanges are mounted on mould walls. This design requires 2 side cores one is in round shape and other one has got the shape of side wall of component. Figure 7, shows the assembly of side cores.

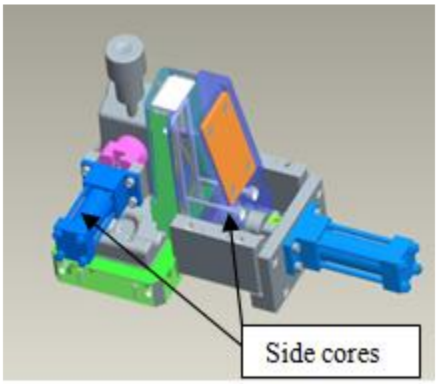

Figure 7. Side cores

VII. CONCLUSION

The work, deals with the designing of injection mould tool of an air filter box bottom cover. The designing was carried out with wildfire pro-e 3.0. 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 The mould 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] Osswald, T., Sheng, L., Gramann, P.J., Injection Molding Handbook. Munich.Hansner (2002)

[2] Hamdy Hassan, Nicolas Regnier, Guy Defaye, A 3D study on the effect of gate location on the cooling of polymer by injection molding, International Journal of Heat and Fluid Flow 30 (2009) 1218–1229.

[3] 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 171 (2006) 259–267.

[4] Technical Directory on Design and Tooling for Plastics, CIPET.

[3] R.G.W. Pai, Injection Mould Design, 2nd Edition.

[4] D. Mathivanan, M. Nouby and R. Vidhya, International Journal of Engineering, Science and Technology Vol. 2, No. 2, 2010, pp. 13-22

[5] J. Nabialek, J. Koszkul Model optimisation for mould Filling analysis with application CAE package C-Mold.

[6] Design analysis and simulation in injection in-mold simulation, Hauptberichter: Prof. Dr.Ing. H.G. Fritz Mitberichter: Prof. Dr.Ing. C. Merten, Institut für Kunststofftechnik der Universität Stuttgart.2009

[7] J.Nabialek, J.Koszkul; Selected methods of modelling of polymer during the injection moulding process; AMME, September 2007

[8] E. Bociga, T. Jaruga, Experimental investigation of polymer flow in injection mould, Archives of Materials Science and Engineering, Volume 28 Issue 3 March 2007, Pages 165-172.