Design and Fabrication of Single-Cavity Injection Mold Using Hot Runners and Direct Hot Tip Gates

Prof. D. N. Inamdar[1]*, Prof. G. V. Chiniwalar[2], Prof. G. A. Naik[3]

[1]Dept. of Mechanical, Hirasugar Institute of Technology, Nidasoshi, Karnataka, India

[2]Dept. of Mechanical, Hirasugar Institute of Technology, Nidasoshi, Karnataka, India

[3]Dept. of Mechanical, Hirasugar Institute of Technology, Nidasohi, Karnataka, India

Abstract – A hot runner system is an assembly of heated components used in plastic injection molds that inject molten plastic into the cavities of the mold compared to a cold runner is simply a pathway or channel formed between the two halves of the mold i.e core half and cavity half for the purpose of carrying plastic from the injection molding machine nozzle to the cavities. Each time the mold opens to eject the newly formed plastic parts; the material in the runner is ejected as a waste. A hot runner system usually includes a heated manifold and a number of heated nozzles. The main task of the manifold is to distribute the plastic entering the mold to the various nozzles which then meter it precisely to the injection points in the cavities. Hot runner technology is continuing to experience a phase of intensive development brought about by the increasing demands of the plastics processing industry. The industry is facing is in need of meet the following requirements. The effect of material, pressure, the temperature of the hot runner nozzle assembly is investigated by ensuring proper manifold and nozzle design. In this the paper proper design of hot runner mold and Plastic flow analysis method is used to optimize the injection position and configure the process parameters rationally so that quality and higher production rate can be achieved as per the customers requirement

Keywords- Hot Runner Nozzle, Hot Runner Technology, Optimal Design, Injection Mold, Hot Tip Gate, Single-Cavity, Manifold Block .

---------------------------♦-----------------------------

I. INTRODUCTION

Cold runner and hot runner systems have been used in injection molds for manufacturing thermoplastic material products used widely in automotive industries and daily used products in day to day life. Cold runner systems offer low-cost mold fabrication, have the potential to apply various gate system structures, and easy modification of mold structures. Despite these advantages, however, the cold runner can result in more loss of raw materials and longer cycle time due to increased cooling time more than the injected products weight. Moreover, restrictions apply to multi-cavity mold systems1. If a cold runner is applied to a multi-cavity mold, high-pressure injection machines are needed because the resin's liquidity decreases with increasing number of fluidic channels, significantly increasing the manufacturing costs. Runnerless injection processes based on a hot runner can be suitable for conserving raw materials, automatizing plastic injection processes, and reducing manufacturing costs. For multi-cavity molds to which cold runner systems cannot be applied, hot runner systems, such as insulated runners, open gate hot runners, and valve gate hot runners, must be used. If these types of runner systems are applied, injection pressure loss could be reduced and the low temperature of the plastic resin could be maintained. This would improve not only the resin properties, thermoplasticity and also the product quality because residual stress near the gates could be reduced. However, gate traces can remain on the upper side of the product.

Moreover, the mold fabrication cost can be reduced by reducing the number of nozzles as well as number of gate points in single cavity mold of complex parts, since a single nozzle can assure the filling of intricate areas of the product as shown in figure 1. However, mold parts can be deformed or destroyed by excessive injection pressure because the wall thickness of cores near the gates is quite small, owing to the characteristics of gate systems. Therefore, verification is required at the design step through computer-aided engineering (CAE) technology. In this article, the entire process of preparing a single-cavity plastic injection molding system with hot tip gates is presented. This process consists of single cavity injection mold design, filling analysis of mold cavity.

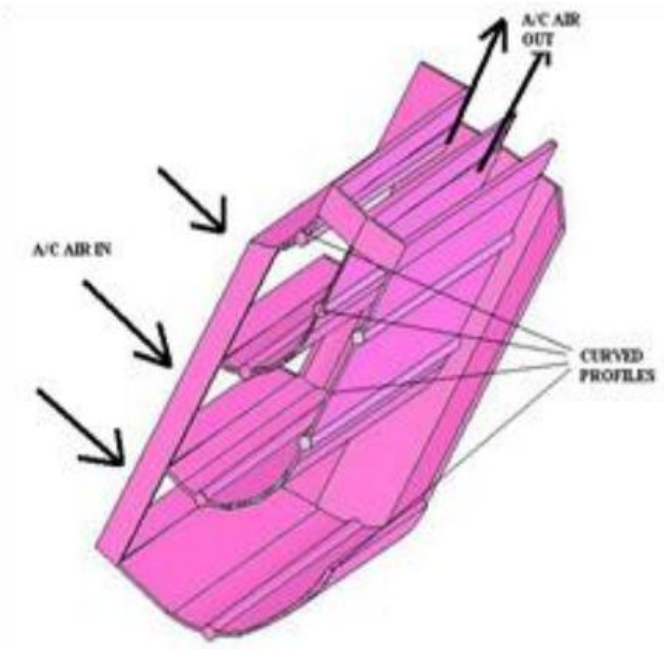

Figure 1 Three dimension model of the A/C SPOILER

II. HOT RUNNER MOLD DEIGN

The main objectives of the research is to design and develop cost effective hot runner unit mould with good design aspects.Manifold and nozzles are designed fabricated in separate units of required shape, enabling flow paths to be shortened. This makes it possible to design hot runner feed system required manifold shape and nozzles. This system gives the user considerable freedom to locate nozzles in the mould using standard manifolds, which lower the cost of the mould.Achieving reliable leak proofing in systems with external heating, one approach to this problem is the concept of the nozzle screwed into the manifold.Reduction in melt temperature fluctuations in systems using microprocessor-controlled heater control units with self-optimization.In this article design of hot runner mold for the part A/C spoiler used in A/C units of cars.The reason for the selection of the hot runner system compared to cold runner system is In cold runner feed system the runner wastage was 30-40% of the component weight.Production rate per hour was less as Cycle Time was too long,because of 4 curved surfaces with 7 flats strips as shown in figure 1.Observing all these designing of cold runner feed system difficulties and wastage of materials and low production rate in existing mold , an attempt is made to select hot runner feed system over cold runner feed sytem

A. Product information.

The product presented in this article is A/c Spoiler used in cars in A/C unit.For this single-cavity hot runner Injection Moulding Tool set has been designed based on various design calculations. The component details are listed in Table 1.

Table 1.Component Details

Material	Poly Propylene (20% Talc filled)
Density	1.10 gm/cm3 = 1.10 x 103 Kg/m3
Shrinkage	1.2 %
Volume	276092.23mm3
Wall thickness	1.5mm
Production quantity	1.5 Lakh per Year
Weight	303.7gms

B. 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, molding sample, a profiled parting plane is chosen. This permits easier ejection of the molding.

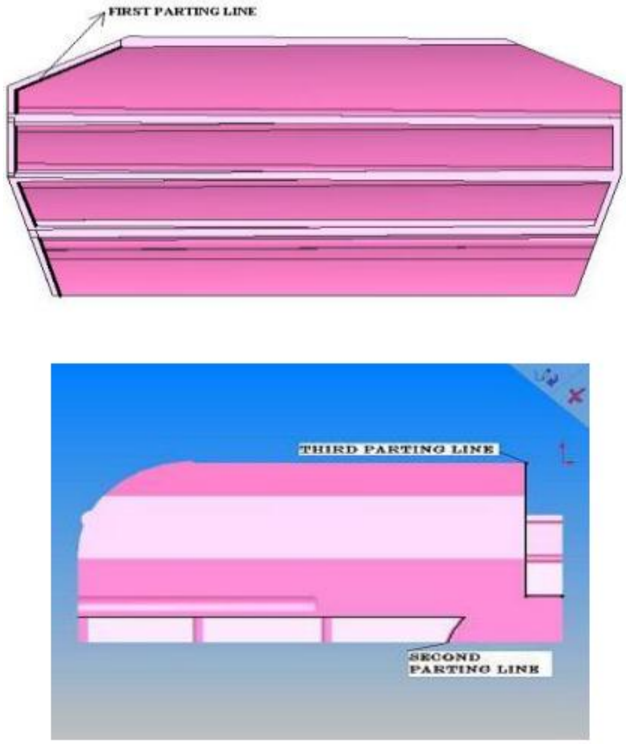

Fig 2. Parting Plane of the component

C. Placement of cavities

As Single-cavity mould is to be designed. The placement of impression is as shown in fig.3. Here the naturally balanced runner has equal distance and runner size from the sprue to the cavity, so that cavity fills under the same condition uniformly.

Prof. D. N. Inamdar[1]*, Prof. G. V. Chiniwalar[2], Prof. G. A. Naik[3]

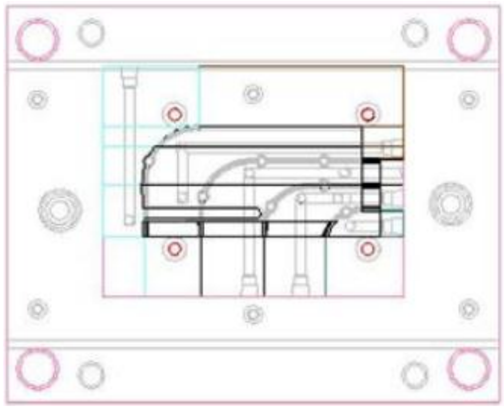

Fig 4. Placement of Cavity

D. Shot capacity of the machine

Shot capacity required in Gms is given by

Rated shot capacity of the particular polymer,

$Ws = Vs \times \rho \times C.$ ……2

Where, Vs=Swept volume (from Machine Specifications)

ρ = 1.1 x 103 Kg/m3 (Density of 20% Talc filled PP)

C: For amorphous 0.93 & for crystalline materials 0.85

Ws =350.6gms < 375gms

E. Plasticizing capacity of the machine

Plasticizing capacity of the machine is calculated using by referring Machine catalogue of Winsor make SP-180 injection molding Machine.

Shot plasticizing capacity required is capacity of 25.8gm/sec

Clamping Force

Clamping force is the force required to keep the mold halves together when injection takes place. Clamping force (CF) can be calculated by the formula

Required Clamping Force C.F = 118.318 tons

Clamping force required is well within the machine capacity of 180 tons.

F. Machine selection

As per the design calulations, SP-180 (Windsor make) Injection molding machine selected. The Clamping Tonnage available on the machine is 180tonnes, Shot weight is 338 gms and Plasticizing capacity is 90 Kg/hr, which are more than the required clamping force, Shot Weight and Plasticizing rate for the mold. So the selected machine is correct.

III. HOT RUNNER NOZZLE/HOT TIP GATE DESIGN [1,2,3,4]

A. Study of the component

- Part name: A/C Spoiler (197x91 x162 mm)
- Material grade: 20% talc filled polypropylene
- Material specific gravity: 1.1
- Manufacturer: PI polymers (India)
- Injection time: 1.26secs
- Injection pressure: 100Mpa
- Shot volume: 271.8cm3/sec (from mold flow analysis)
- Weight of the component: 318.8gm
- Mold Temperature: 55°C
- Material Temperature: 250°C
- Flow rate for 4 nozzles in cu.cm: = 53.9cm3/sec

B. Slection of Nozzle and Heater Capacity [1,2,3,4]

Based on the flow rate 53.9cm3/sec hot runner nozzle sizes are decided.Nozzle length and inner through hole diameter is taken from the graph shown in figure 5 below for the material polypropylene.

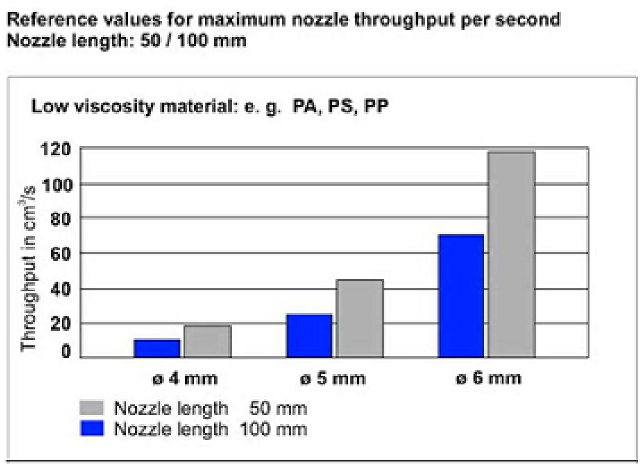

Fig.5 Graph of Selection Nozzle Length

Prof. D. N. Inamdar[1]*, Prof. G. V. Chiniwalar[2], Prof. G. A. Naik[3]

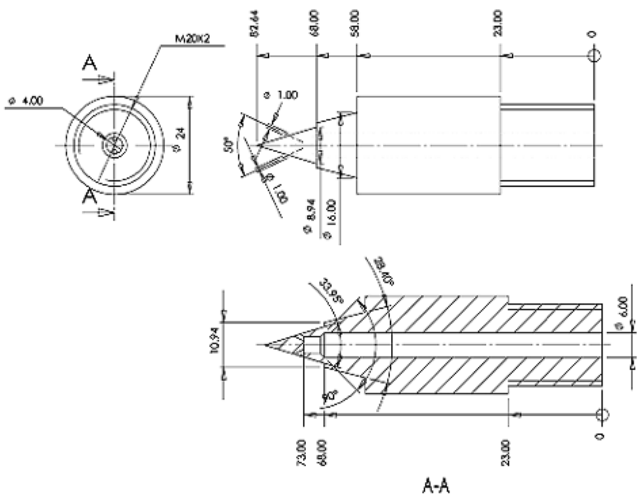

Fig 6. Schematic view of Nozzle

C. Heater slection for heated manifold [2,.3,4]

The most important is selection of heating elements for smooth running of hot runner mould. Their use depends primarily on the required wattage and the available space.

The highest wattage in the smallest space can be provided with high efficiency cartridge heaters. So cartridge heaters are used to heat the manifold.

The Total Power Output to be installed can calculated for the manifold from the equation

P=(0.115θm)/(860T□)kW where, m= mass of the manifold

θ = temperature differential between decided melt temperature and manifold temperature at the onset of heating.

i.e Tmatl=Melt temperature in deg C,Tmould=Mold temperature in deg C.

T = Hrs of heating)

η = total efficiency (electrical-thermal)

Power required is 1.0416kW

So 6 cartridge heaters of same size of 250watts each can selected.These Heaters are placed evenly around the nozzle location as per drawing in the holes drilled for their location on the hot runner manifold.

D. Heater Selection for Nozzles [1,2,3,4]

Coil heaters have been used in this hot runner system to greatly increase the cycle speed and the quality of the product produced. Due to an even heat around the nozzle body, the material is not subject to excessive temperature changes, resulting in less degradation of material.

Calculation of wattage required for coil heaters is done using the equation 4

P= (0.115θm) / (860T□)kW…….5

m= mass of the Nozzle

θ = temperature differential between decided melt temperature and manifold temperature at the onset of heating.

i.e Melt temperature in deg C, Mold temperature in degC.

T = Hrs of heating

η = total efficiency (electrical-thermal)

So minimum power required is 54watts.

Hence coil Heaters of 75watts are selected to compensate heat loss and increase life of heaters.

G. Runner Design [1,2,3,4]

Runner Type: In this hot runner design full round runner is employed to feed the component. Because in hot runner system round runner is used to reach out the molten plastic upto nozzle with less shear and with low pressure drop and with higher efficiency and also its ease of manufacturability.

No matter how the nozzles are laid out in the hot runner mold, only continous melt can guarantee the product quality.

The volumetric velocity of melt flow Q is calculated as:

$$Q = W/s\rho_m\,(cm^3/sec) \ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots 6$$

Where, = weight of the component in (gms)

s = Injection time in (sec) (from plastic flow advisor)

ρ_m = Melt density (g/cm) (from plastic flow advisor)

$$Q = 271.8\,cm^3/sec$$

Runner diameter is calculated using the following equation

Where, n = non Newtonian Index (/sec), = 24000/sec

Q = Volumetric velocity of the melt in the main runner,

Prof. D. N. Inamdar[1]*, Prof. G. V. Chiniwalar[2], Prof. G. A. Naik[3]

γ_s = Shearing velocity of the plastic melt in the main runner and is usually taken as 5 x 10 /sec,

π = internal Pressure in (Mpa), it is half the maximum pressure of the machine.

Runner diameter = 6.8mm .

So selectd diameter of the runner is 10mm to overcome the flow losses.

H. Gate Design.[1,2,3,4]

Gate type selected for the component is based the type of material used and the gate location on the component.

Material used is 20% talc filled polypropylene, a. crystalline material.Gate location is located using proe-2 plastic flow advisor.

Gate type selected is hot tip gating or thermal gate.

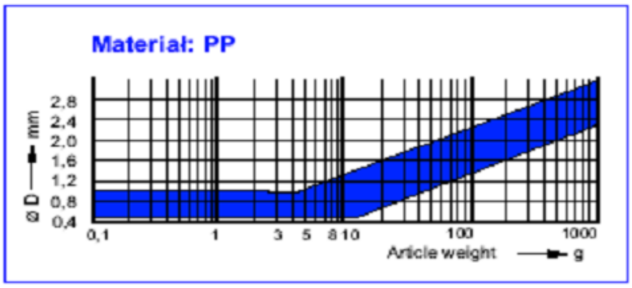

Fig 7.Selection Gate diameter Vs Article weight

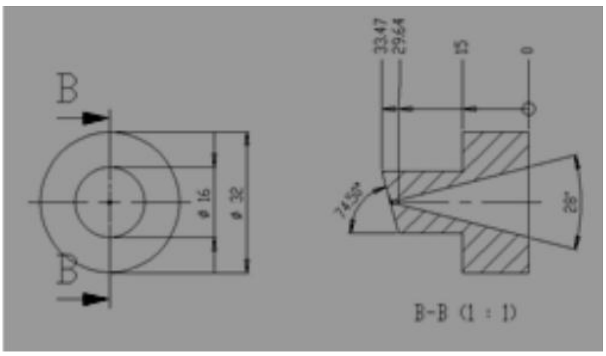

Fig 8 Schematic diagram of Hot tip gate

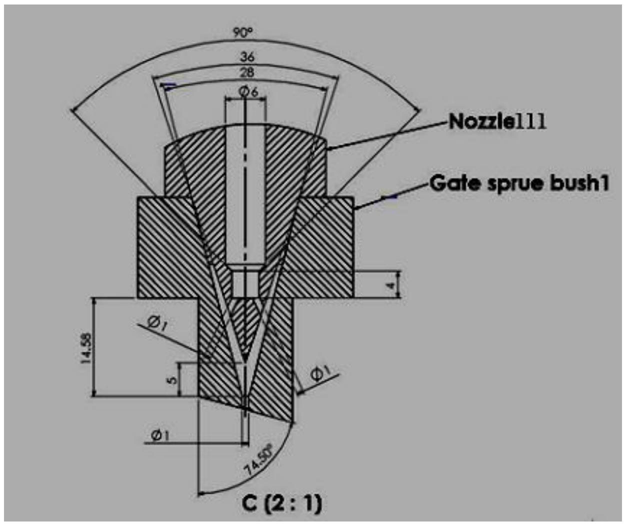

Fig 9. schematic view of Nozzle & Gate assembly

I. Solidifying Time

The solidifying time is proportional to square of the wall thickness.

Solidifying Time T = ρ ×a ×t²/8 Tcp (Tmatl – Tmould)

Where, ρ = Density of the plastic material (PP 20% talc filled) ,a = Heat content of PP 20% talc filled material

t =Wall thickness component,

Tcp = Thermal conductivity of the 20% TALC FILLED PP in ^{0}K

Tmatl = Injection temperature of the melt in 0K (Table 2.3)

Tmould = Temperature of the mould in □K (Refer Table 2.3)

Solidifying time (T) = 11.4 sec

J. Cycle Time Estimation

Cycle time = Fill time + solidifying time + Mold opening and closing time + Ejection time ……. 9

Cycle time (Ct) = 8 + 11.4 + 32 + 5 = 56.4sec

No. of shots /sec =Total time /Total cycle time= 64 shots / hr.

K. Design of Cooling System [9, 10,]

For this design rectangular type cooling circuit is chosen as shown in Fig 4.5 because it is the most efficient cooling for this design. Cooling hole diameter chosen is 8mm.

Prof. D. N. Inamdar[1]*, Prof. G. V. Chiniwalar[2], Prof. G. A. Naik[3]

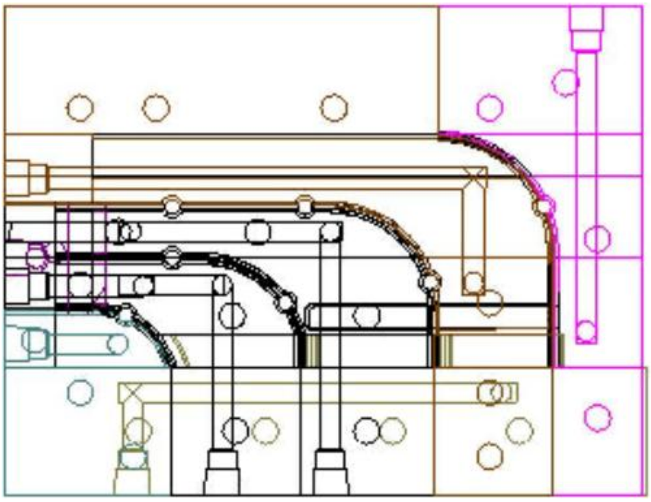

Fig 10. Cooling Circuit

Rate of heat to be extracted by water in J/kg.

Qw = shot weight x heat factor for PP x no of shots / sec.

So Rate of heat extracted is = 3071.26 J/sec = 3071 J/sec

Heat to be transferred per hour by the cooling system.

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 and radiation.

Heat to be removed, Q1= Qw/2=3071 / 2 = 1535.5 J/sec.

Weight of the water to be circulated (Mw)

Mw = Q1 /S [Toutlet – Tinlet] ×K

Where,Q1= Heat to be removed by water = 1535.5 J/sec

S = Specific heat of water = 4.184 J/Kg K

Toutlet – Tinlet= 5 ºC=5+273=278 ºK

K = 0.65 for direct cooling

Mass of the water to circulated (Mw) =7308 liters/hr.

IV. PLASTIC FLOW ANALYSIS

Plastic flow analysis is carried out using the PLASTIC ADVISER of "Pro-Engineer Software." of Parametric Technology Corporation.

A. Introduction to Plastic Advisor [7]

Plastic Advisor analysis method is used to optimize the injection position and configure the process parameters rationally so that the molding of quality product by minimizing the weld lines, air traps etc,and also analyze confidence of fill, injection time, cycle time ,difference in flow front temperature etc.uniformity in wall thicknesscan be maintained . The evaluation and optimization method using Plastic Advisor analysis method provides a reliable technical method for improving the quality of injection mold with best location of gates.The input parametes used are polymer and injection location. Other settings such as material temperature, mould temperature, pressure, are selected automatically according to the material properties.

First, using Pro/E software to achieve products three dimension modeling and mold assemble model which are shown in Figure1 and Figure2 respectively, then the gating system design, select the shape of hot tip gate, sub channel and cross section of gate as round according to relevant information.

After the analysis the following results have been observed

Confidence of fill,Fill time,Injection pressure,Pressure drop.

Flow front temperature,Expected quality of the part,The weld lines and Air traps formation.

The results are shown as color shaded pictures, color plots. These are set norms for acceptability of results of these analyses. If the results of the analysis are within set norms, then the part and mould design becomes acceptable. Also we can review the inputs and try to resolve the molding problems. Otherwise the designer has to modify the part design or mould design to overcome the constraints visible in the analysis.

Before analyzing the above results it is required to know the polymer injection location to the cavity.

Inputs to Plastic Advisor

Material	: 20% talc filled Polypropylene
Injection Pressure	: 100 Mpa
Trade Name	: Acctuf 3247w
Melt Temperature	: 240°C
Manufacturer	: BP Chemicals
Mold Temperature	: 55°C

B. BEST GATE LOCATION RESULT

When one or more injection locations already exist, the result suggests the best place for the next gate location given the selected material properties. It then rates the model areas for their suitability for an

Prof. D. N. Inamdar[1]*, Prof. G. V. Chiniwalar[2], Prof. G. A. Naik[3]

injection location where the worst position is classified as the least suitable and the best position is classified as the most suitable.

The gate location examines these four aspects of the part.

1. Processability.

2. Minimum Pressure (lower shear rates).

3. Geometric Resistance (Over packing).

4. Thickness.

The gate is located on one of the highly recommended area (yellowish Blue color) as shown in fig. 11.

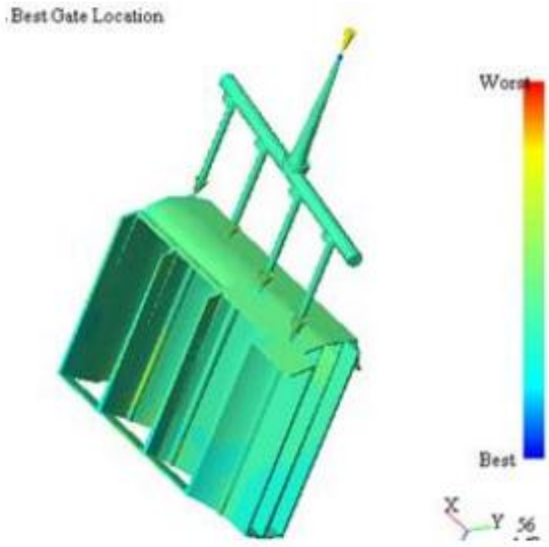

Fig 11: Best gate location selection

C. The derived results from analysis confidence of fill

Processing parameters

Injection pressure	-100 Mpa
Recommended Melt Temperature	-240°C
Recommended Minimum Mold Temperature	- 55°C

Fig 12.shows the plot of confidence of fill.

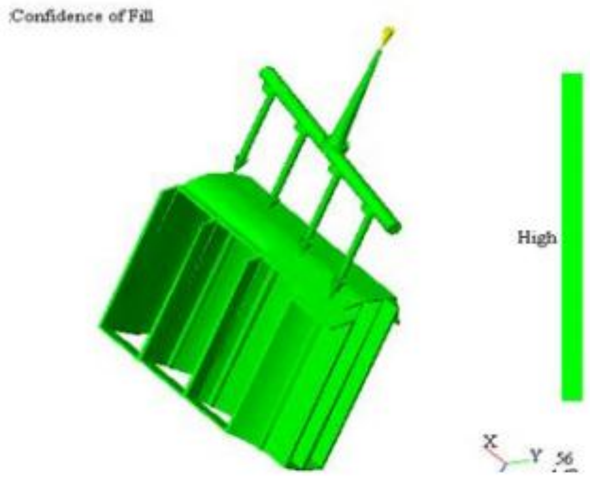

Fig 12: Confidence of Fill

D. Fill Time

The fig 13 shows the plot of fill time prediction. The component has a fill time of 1.26 seconds.

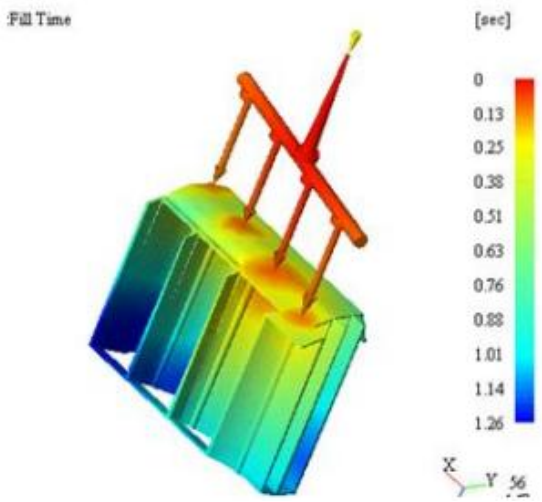

Fig 13: Fill Time

E. Injection Pressure

The fig 14 shows the plot of injection pressure. The injection pressure for selected gate location is 8.74Mpa.

Prof. D. N. Inamdar[1]*, Prof. G. V. Chiniwalar[2], Prof. G. A. Naik[3]

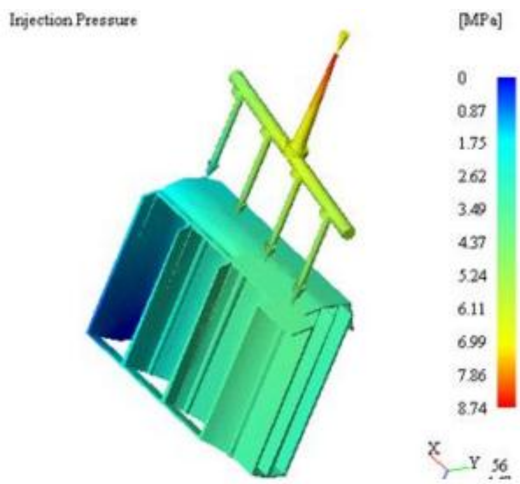

Fig 14: Injection Pressure

F. Pressure drop

The pressure drop is one factor used to determine the confidence of fill result. If the pressure drop is greater than 80% of the current injection pressure setting, then it causes a yellow confidence of fill. Figure 15 shows the pressure drop which can be acceptable.

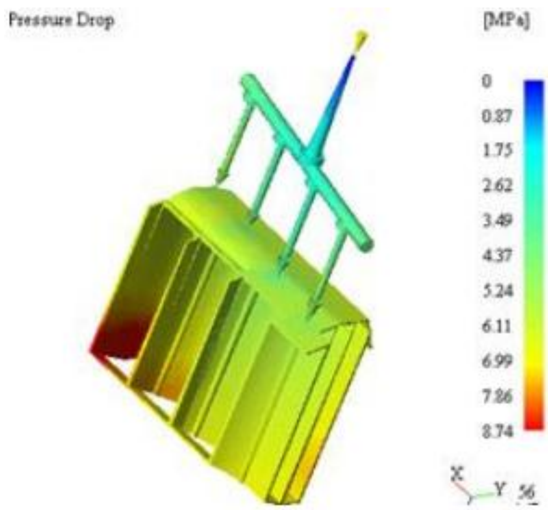

Fig 15: Pressure Drop

G. Flow Front Temperature

If the flow front temperature is too low in a thin area of the part, hesitation or short shot may occur. In areas where the flow front temperature is too high, material degradation and surface defects may occur. Make sure that the flow front temperature is always within the recommended temperature range for the polymer we are using. The figure 16. Shows the difference in temperature at the flow front was less than 27.120C. This indicated that an injection time of 8.76secs was the most suitable.

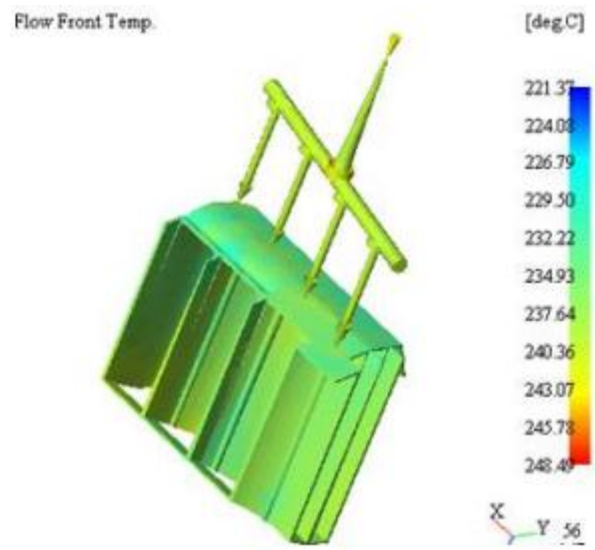

Fig 16: Flow Front Temperature

H. Expected Quality

The quality result measures the expected quality . This is derived from the pressure, temperature and other result such as flow front temperature, pressure drop, cooling time, shear rate and shear stress. For each area of the cavity, the five results are evaluated. If all the five results in an area are acceptable the area is green. If there is at least one unacceptable result the area is red.Fig 17 shows the expected quality of component.

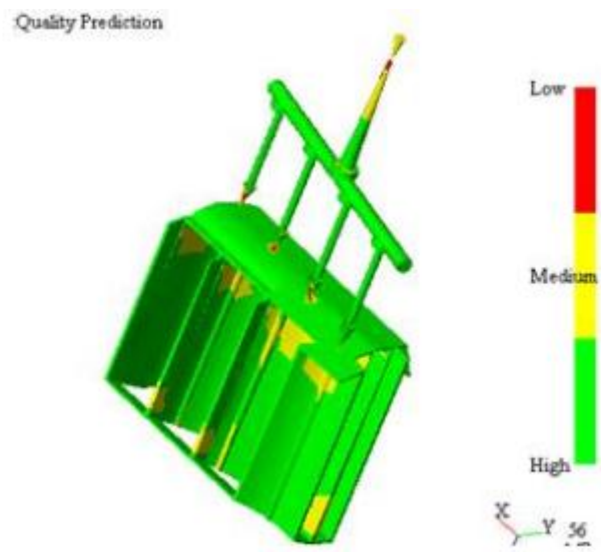

Fig 17: Expected Quality of the Component

I. Weld lines

A good weld occurs when the melt temperature is no lower than 200C below the injection temperature.The fig 18 shows the plot of weld lines formation (Red color) on the part.

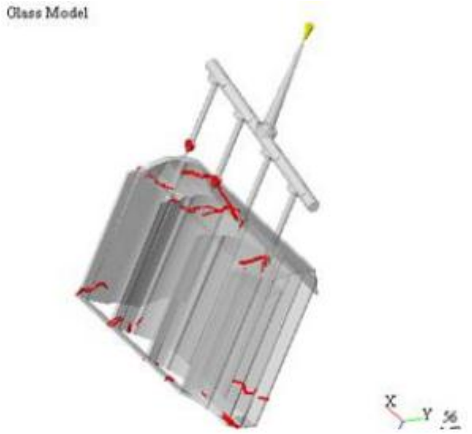

Fig 18: Weld Line formation

J. Air Traps

The air trap results shows the regions where the melt stops at convergence of at least two flow fronts or at the last point of fill, where a bubble of air becomes trapped. Air traps can occur when the converging flow fronts surround and trap a bubble of air. This normally happens where there are unbalanced flow paths. The figure17 shows the air traps at end portions of the component which are negligible as per functional requirement of the component.

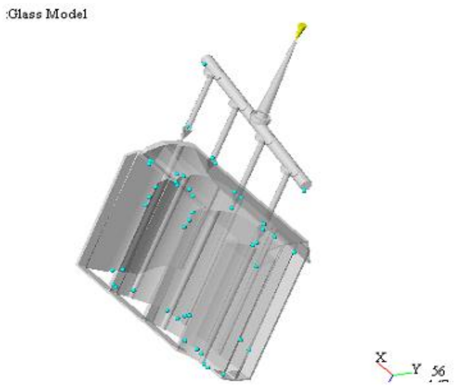

Fig 19: Air Trap

V. 3D SOLID MODELS OF THE TOOL IN VARIOUS VIEWS

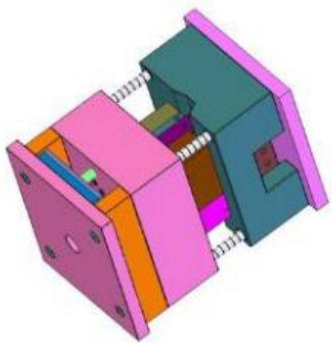

Fig. 20. Mold Tool in Closed condition

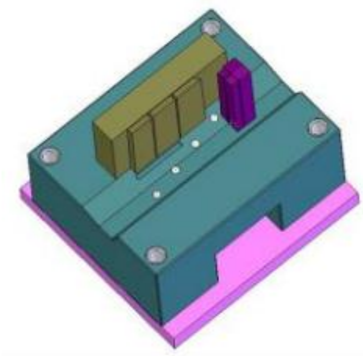

Fig. 21 Fixed half or cavity half of the Tool

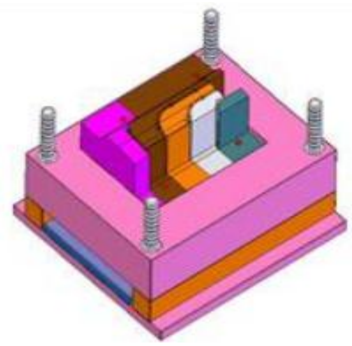

Fig.22 Moving Half OR core half of the tool

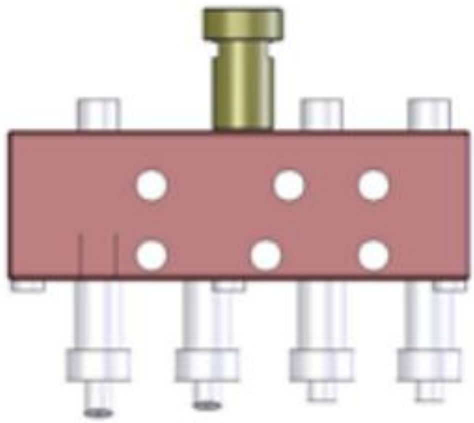

Fig.23 Hot runner manifold

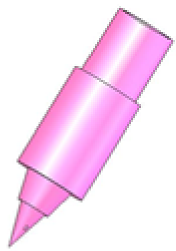

Fig.24 Nozzle

Prof. D. N. Inamdar[1]*, Prof. G. V. Chiniwalar[2], Prof. G. A. Naik[3]

Fig .25 Gate Sprue Bush

REFERENCES

"Gunter hot runner Technology" technical information manual

"DME hot runner technology" technical information manual

"Mastip hot runner technology" technical catalogue

Jianliang Lai, Shuyou Zhang, Jianrong Tan and Chunwei Xu "Knowledge Based Research on Integrated Design Methods of Hot-Runner Technology for Multi-Cavity Injection Molding" Proceedings of the 2007 IEEE International Conference on Mechatronics and Automation,August 5 - 8, 2007, Harbin, China

A. Demirer , Y. Soydan , A.O. Kapti" An experimental investigation of the effects of hot runner system on injection molding process in comparison with conventional runner system" , Elseveir,Materials and Design 28 (2007) 1467–1476

Tae-Il Seo1, Byeong-Uk Song1 and Jeong-Won Lee2,"Design and fabrication of multi-cavity injection mold using hot runners and direct edge gates",SAGE Publications ,Institution of MechanicalEngineering Science ,2014, Vol.228(15) 2790–2798 IMechE 2014

Zhou Chun-ying,Wang Li-tao," Injection Mold Design based on Plastic

Advisor Analysis Software in Pro/E", 2011 International Conference on Multimedia and Signal Processing

Corresponding Author

Prof. D. N. Inamdar

Dept. of Mechanical, Hirasugar Institute of Technology, Nidasohi, Karnataka, India

E-Mail – sudarshan.inamdar@gmail.com