International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 7 Issue 03, March-2018

A Simulation Study of Gates and Cooling Systems in Plastic Injection Molding

Shaikh Mohammed Ishak*, Shaikh Mohammed Nabil**, Sheliya Mujahid Ilyas***
B.E. Students, Department of Mechanical Engineering, Rizvi College of Engineering
Mumbai-400050, India

***Abstract*—Plastic injection molding processes are used to produce plastic parts and components, which finds applications in many industrial as well as household consumer products. Plastic injection molding is really a challenging process for designers, researchers, and manufactures to produce the products at low cost, meeting all the necessary requirements from the customers. To remain competitive in market, many product designing, mould designing aspects as well as large number of process parameters need to be optimized.**

Autodesk Moldflow is a powerful simulation tool to study behavior of different parameters on final quality of the product. This paper emphasizes on optimizing gate location, type of gate and cooling system for our product (Switch Board Frame). This paper describes the influence of gate location, type, and size through a repeated number of analyses which is carried out by software aforementioned to reduce fill time, scrap and automatic degating. The process parameters like fill time, pressure drop, quality, weld lines, sink marks, and air traps are analyzed by simulation in successive trials. This paper also presents a simulation study of different types of cooling systems and compares the performance in terms of volume shrinkage, temperature variance, frozen fraction, and cooling quality to determine which configuration is more appropriate to provide uniform cooling with minimum cycle time.

Keywords— Air traps; Autodesk Moldflow; Injection molding; Shrinkage; Warpage

I.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 can be molded into various forms and hardened for commercial use. Plastic is perfect for this modern age. It is light, strong, easily molded and durable. Plastic injection molding is really a challenging process for designers, researchers, and manufactures to produce the products at low cost, meeting all the necessary requirements from the customers. To remain competitive in market, many product designing, mould designing aspects as well as large number of process parameters need to be optimized.

The typical process cycle time in injection molding machine varies from several seconds to tens of seconds depending on the part weight, part thickness, material properties and the machine settings specific to a given process. Process control of injection molding has a direct impact on the final part quality and the economics of the process. In the injection molding processes, gate location and its type is very important design parameter which is in the relation with polymer capability, part shape and dimension, mould structure and mould condition, also influences the manner in which plastic flows in to the mould cavity. In the cycle time the cooling time can represent more than 60% of the injection cycle. Cutting down the cycle time for each part is a major concern in injection molding machine. In order to set the processing parameters, they commonly follow on experience and trial-and–error method. This process becomes inadequate and unpractical for complex products. As a consequence the designers need a more powerful tool to analyze and to optimize the process. So, the motive of this paper is to determine the ideal parameters for our product (Switch Board Frame).

II.PART DETAILS

Material: PC (Polycarbonate)

Chemical Formulae: $C_{15}H_{16}O_2$

Colour: White

Weight: 26.9 grams

Specific Gravity: 1.19

Wall Thickness: 2 mm

Volume: 22513.02 mm^3

Surface Area: 23464 mm^2

Heat content: 111.1 cal/g

Melt Temperature: 310^0C

Mold Temperature: 95^0C

Polycarbonate is an amorphous thermal plastic material whose high heat resistance and excellent physical properties make it an ideal material for enclosures. Polycarbonate can withstand a wide range of temperature fluctuations and its good electrical properties are unaffected by humidity. As a self-extinguishing material, polycarbonate requires no protective coating.

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 7 Issue 03, March-2018

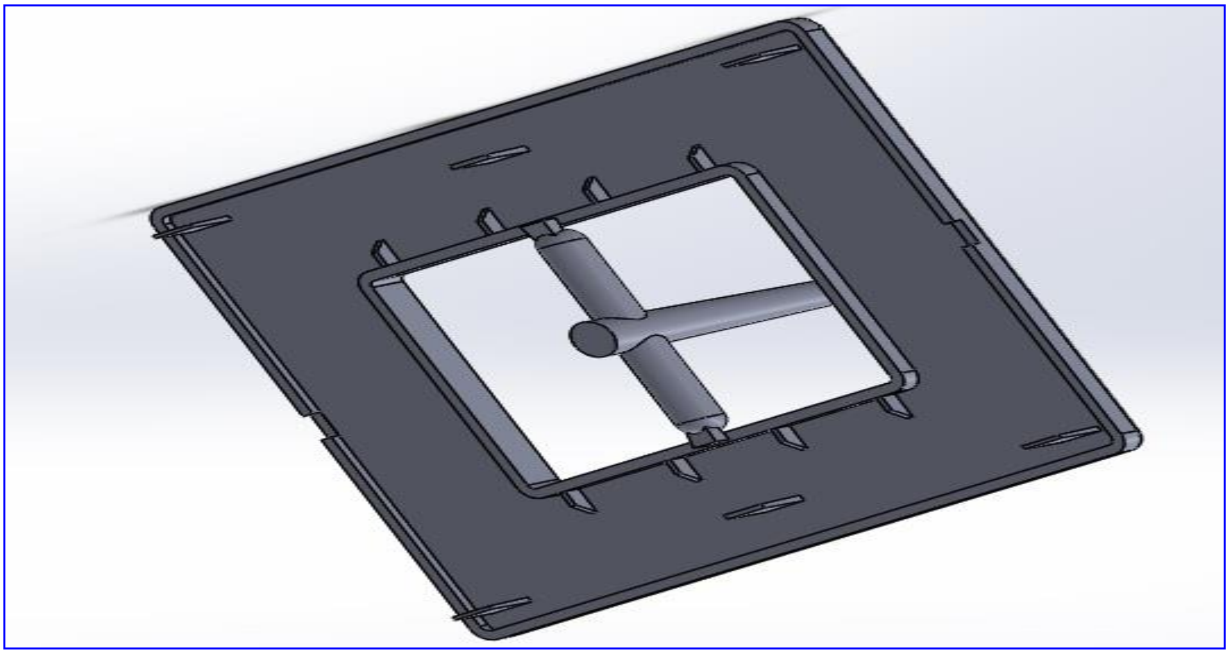


Figure 1: Switch Board Frame

III.PROPOSED METHODOLOGY

The CAD model of our product is made from Solid works. To do further analysis Autodesk moldflow software is used. This software is capable of doing all kind of plastic analysis which is essential for final quality of product. Process parameters such as plastic material, injection pressure limit, etc. and feeding system such as gate, runner and sprue are fixed as it is of prime importance.

In this paper, firstly the ideal gate location as suggested by Autodesk moldflow adviser is determined. There cannot be one gate suitable for all type of plastics, so a good judgment is to be made while selecting a particular type of gate for a plastic product. There will be five trials done for different gates and the one which is best for this application is selected.

This paper also includes cooling system analysis which has a direct impact on cycle time and product quality.

So, for this product Autodesk moldflow adviser has suggested dual domain meshing as the part is thin and is of simple rectangular geometry. There will be simulation study of three cooling systems done and the one which gives satisfactory results is selected.

IV.DETERMINING GATE LOCATION

For determining the ideal gate location with the help of Autodesk Moldflow advisor, the basic part details are required. The gate location examines these five aspects of the part:-

1. Process Ability
2. Minimum Pressure
3. Geometric Resistance
4. Thickness
5. Flow Resistance Areas

So, once the input details are given, the advisor suggests the ideal gate location for the product. It is observed that gate location should be there at thicker sections of the component as it can be seen from Fig. 2.

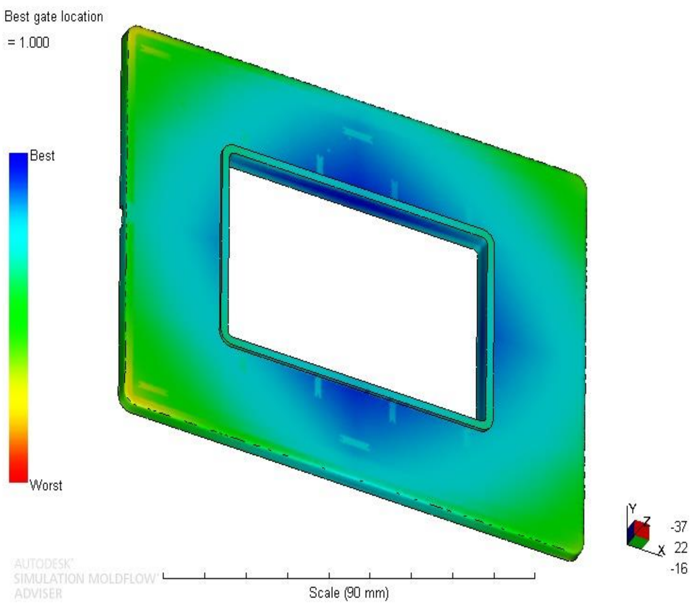


Figure 2: Best Gate Location

V.GATE SELECTION

There are various parameters involved in the result provided by the Autodesk moldflow software which enables us to do trial and error analysis for different input parameters:-

1. Filling Analysis: The filling analysis result displays the probability of proper filling within the cavity. This result is derived from the pressure and temperature results. If the cavity does not fill (short shot) changes must be made to the analysis

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 7 Issue 03, March-2018

like gate location, processing conditions. However, to ensure the finished and good quality product, the cavity must also be adequately packed with plastic. The confidence of fill result can display green, yellow, red and translucent sections of the mould.

2. Fill Time: The fill time displays the probability of proper filling within the cavity. This result shows the flow path of the plastic through the part by plotting contours which join regions filling at the same time. These contours are displayed in the range of colors from blue to indicate the first region to fill, through to red to indicate the last region to fill. A short shot is a part of the model that did not fill and will be displayed as translucent. By plotting these contours in time sequence, the impression is given of plastic actually flowing into the mould.

3. Air Traps: The air traps result is generated in flow analysis, and the graph shows pink dots wherever an air trap is likely to occur. This is eliminated by providing suitable air vents in the mold.

4. Sink Marks: Sink marks are caused by a differential in volumetric shrinkage. They typically occur on the opposite sides of surfaces to which ribs or bosses are attached, and in significantly thicker areas of parts where the volumetric shrinkage is not adequately compensated during the packing phase.

5. Weld Lines: When a weld line forms, the thin frozen layers at the front of each flow path meet, melt, and then freeze again with the rest of the plastic. The orientation of the plastic at the weld is therefore perpendicular to the flow path. The presence of weld and meld lines may indicate a weakness or blemish. It's difficult to avoid weld lines completely; so formation of weld lines is acceptable.

The moldflow simulation is done repeatedly for five different types of gates and the one which gives best results for this product will be selected at the end. The parameters that will be looked upon to decide the best gate are:-

1. Fill Time
2. Injection Pressure
3. Quality
4. Sink Marks
5. Air Traps
6. Weld Lines

1. Edge Gate (Circular Runner)

It can be observed from Fig.3 that for this gate, the fill time is 1.44 s and the injection pressure required is 35.56 MPa. The component is of low quality but there are some longer weld lines of maximum value 18.6 mm. There are certain spots of air traps and magnitude of maximum sink mark in the component is 0.018 mm (shown by red dot) although its maximum value is shown as 0.0342 mm in Fig.3. However, 0.0342 mm will not be considered as maximum sink value as this value is observed in feeding parts which are going to be removed from final product. So, now onwards whenever maximum sink value is specified it is for the final product and not for the entire part.

2. Edge Gate (Trapezoidal Runner)

It can be observed from Fig.4 that for this gate, the fill time is 1.492 s and injection pressure is 36.26 MPa. The product is of good quality and magnitude of maximum sink mark is 0.0152 mm. There are certain air traps and longest weld line is 15.7 mm.

3. Submarine Gate

It can be observed from Fig. 5 for this gate, the fill time is 1.476 s and the injection pressure is 38.4 MPa. The product is of low quality and magnitude of maximum sink mark is0.0181 mm. There are relatively more air traps with longest weld line of 18 mm.

4. Fan Gate

It can be observed from Fig. 6 for this gate, the fill time is 1.351 s and the injection pressure is 35.97 MPa. There are certain air traps and the longest weld line is 15.87 mm. The maximum sink mark is 0.0165 mm and although the product is of good quality, it is associated with problems faced while its degating is done.

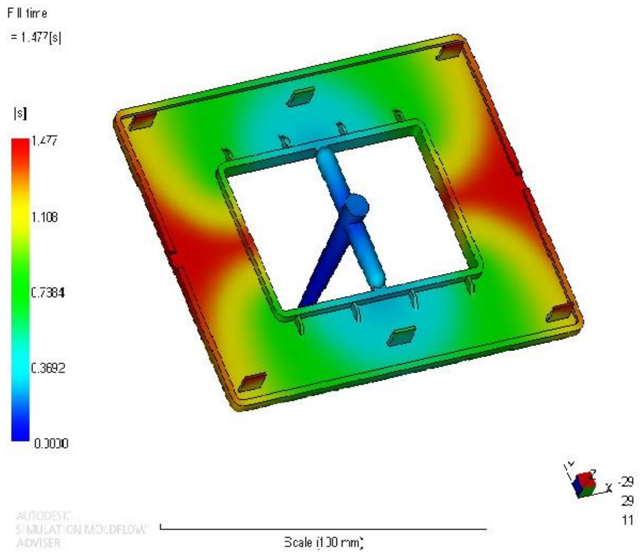
5. Overlap Gate

It can be observed from Fig. 7 that for this gate, the fill time is 1.354 s and the injection pressure is 35.15 MPa. There are relatively less air traps and longest weld line is 16.26 mm. The maximum sink mark is 0.013 mm and the product is of acceptable quality.

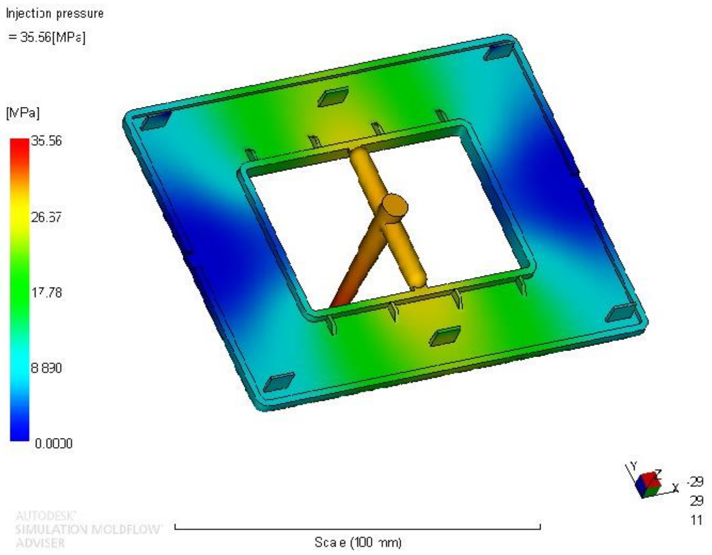
Type of Gate	Edge Gate(Circular Runner)	Edge Gate(Trapezoidal Runner)	Submarine Gate	Fan Gate	Overlap Gate
Fill Time	1.477 s	1.492 s	1.476 s	1.351 s	1.354 s
Injection Pressure	35.56 MPa	36.26 MPa	38.4 MPa	35.97 Mpa	35.15 MPa
Quality	Low	Good	Low	Good	Good
Sink Marks	0.018 mm	0.0152 mm	0.0181 mm	0.0165 mm	0.013 mm
Air Traps	Acceptable	Acceptable	Not Acceptable	Acceptable	Acceptable
Weld Lines	18.6 mm	15.7 mm	18 mm	15.87 mm	16.26 mm

Table 1: Types of Gates

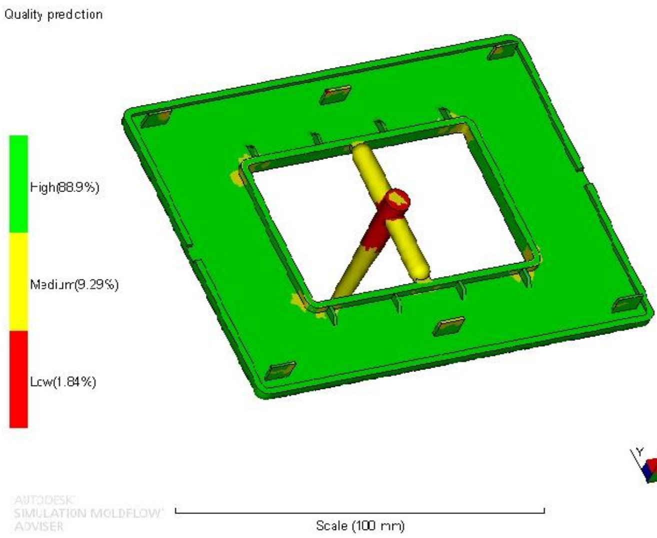
International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 7 Issue 03, March-2018



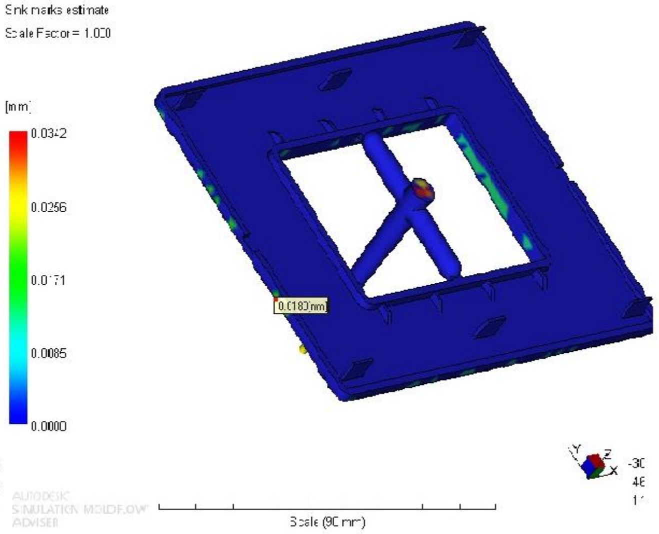
a) Fill Time



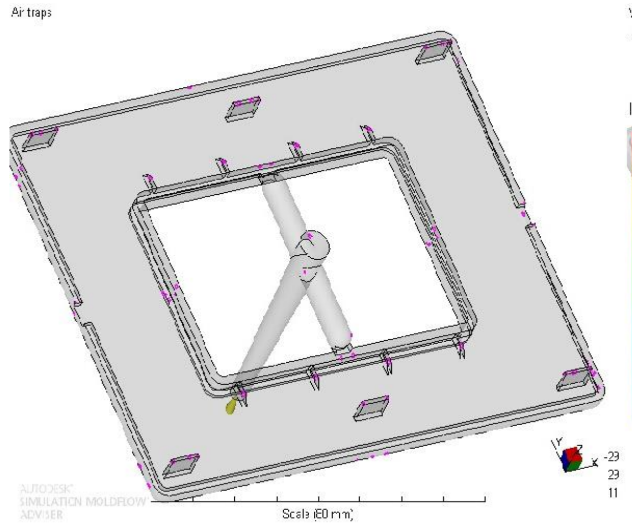
b) Injection Pressure



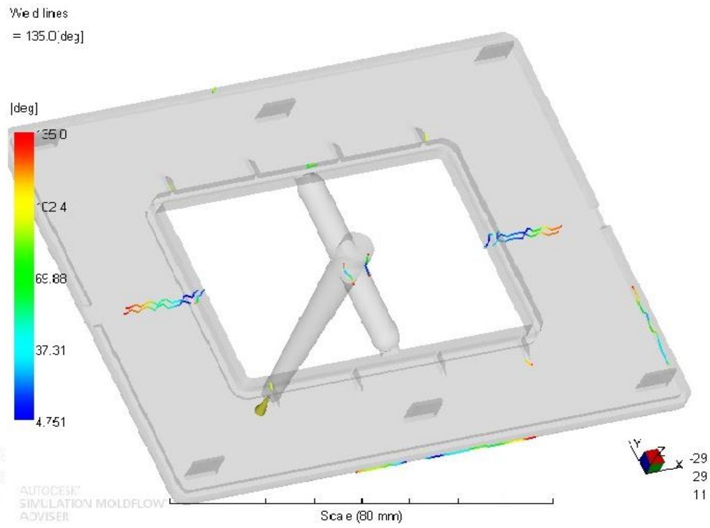
c) Quality



d) Sink Marks



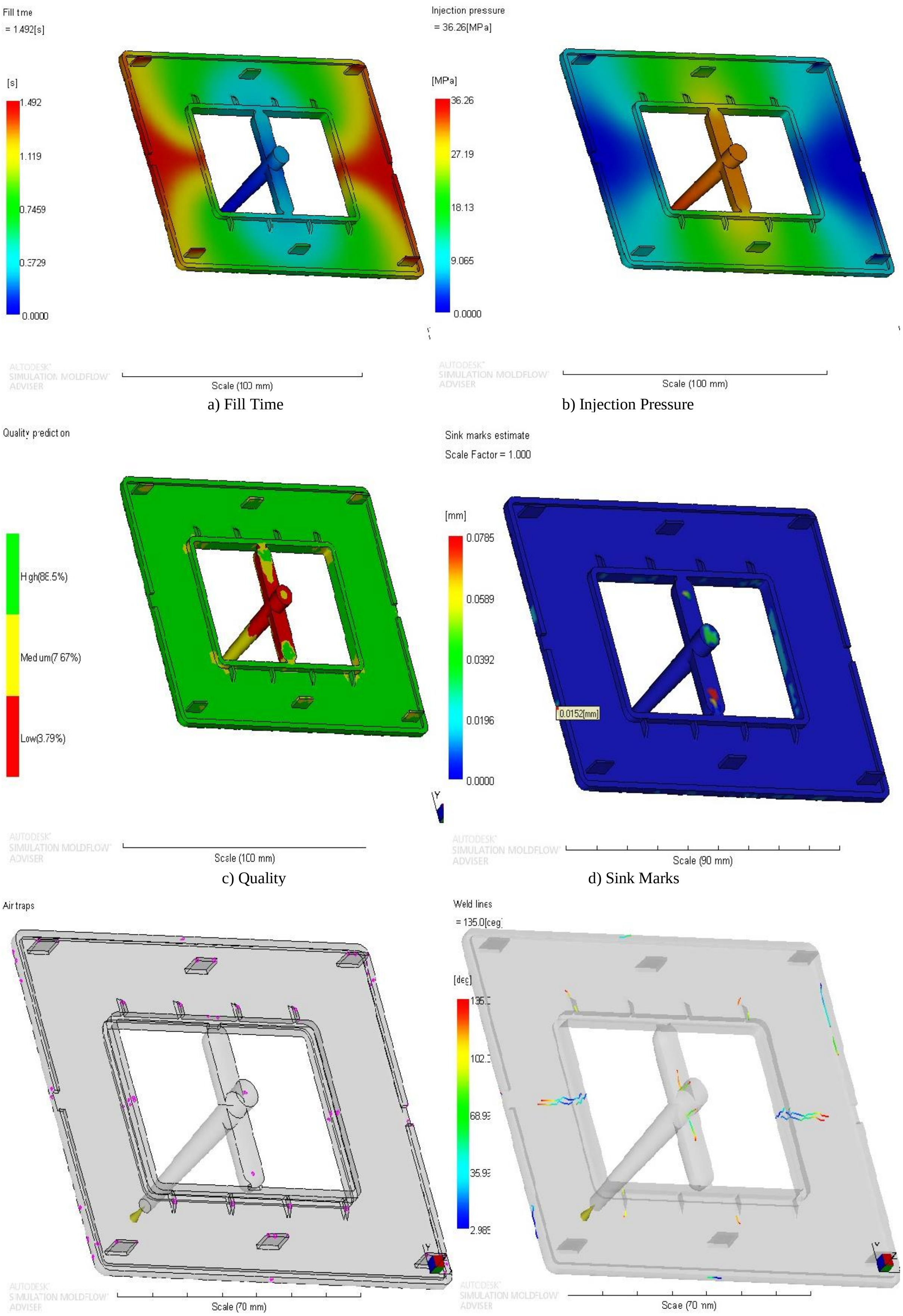
e) Air Traps



f) Weld Lines

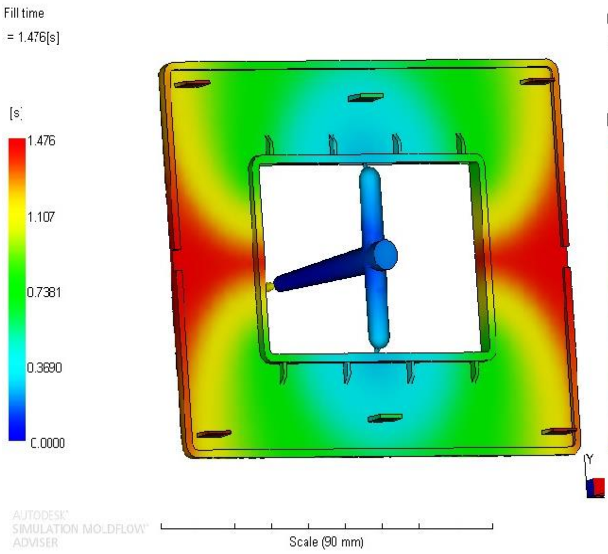
Figure 3: Edge Gate (Circular Runner)

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 7 Issue 03, March-2018

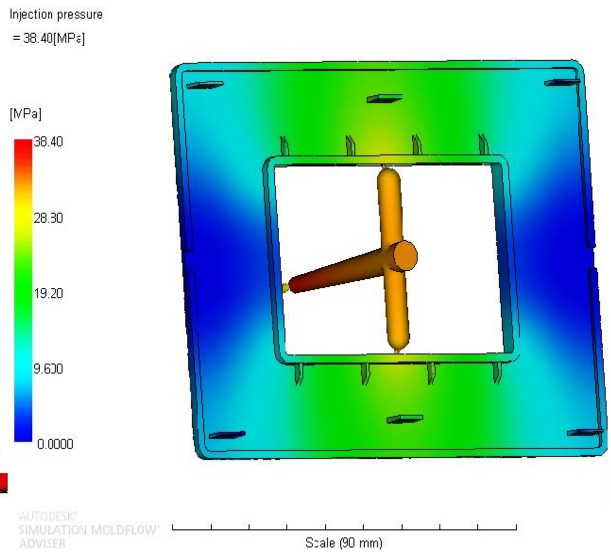


a) Fill Time
b) Injection Pressure
c) Quality
d) Sink Marks
e) Air Traps
f) Weld Lines

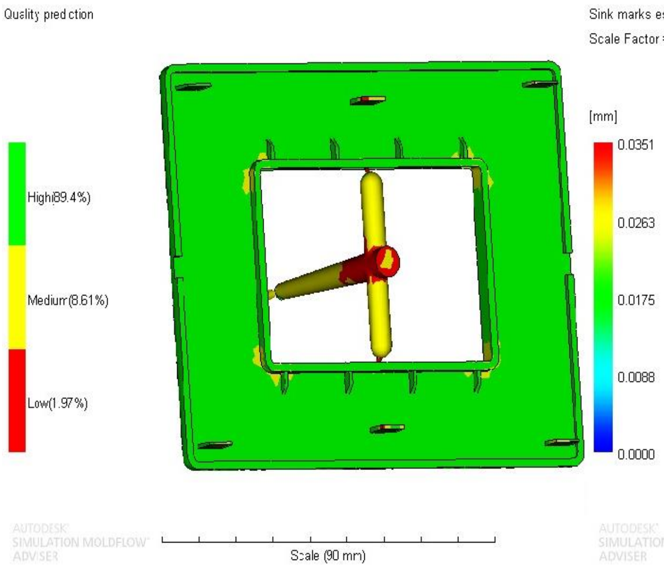
Figure 4: Edge Gate (Trapezoidal Runner)



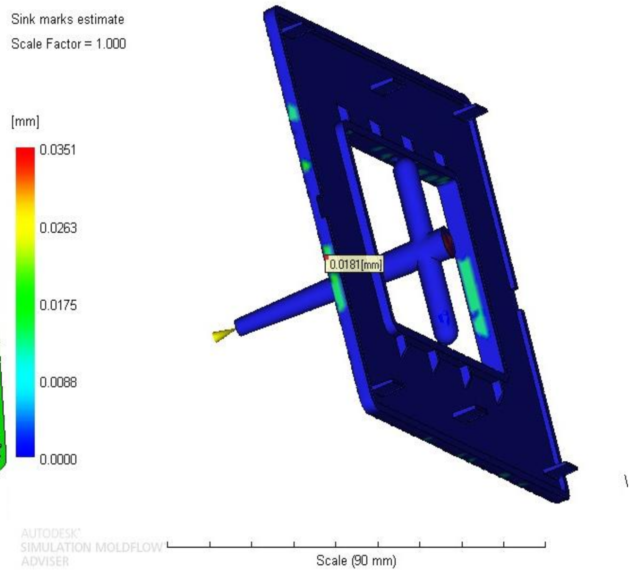
a) Fill Time



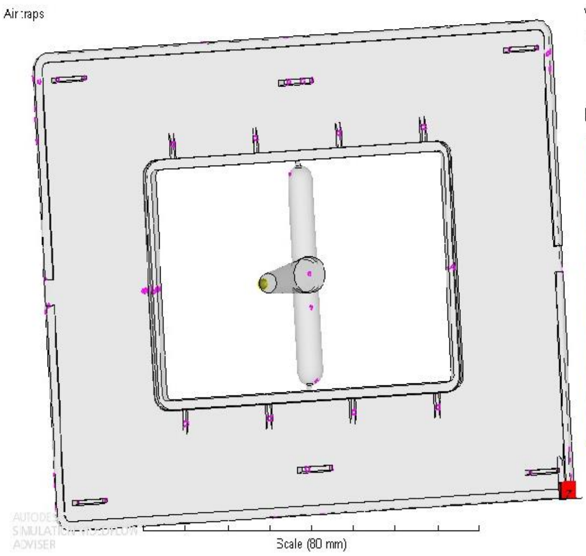
b) Injection Pressure



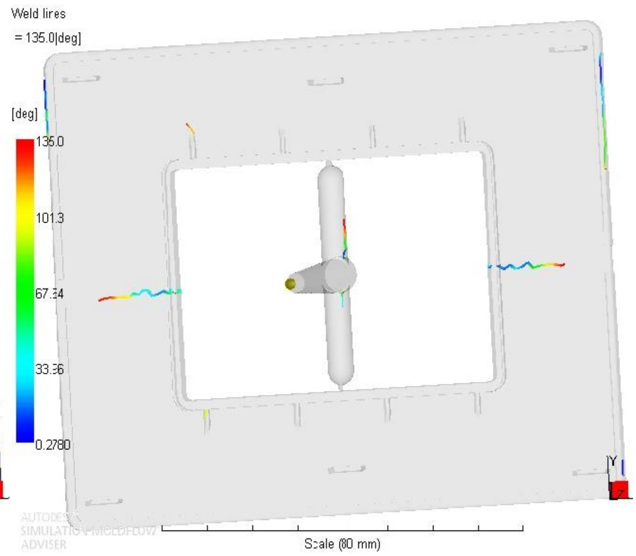
c) Quality



d) Sink Marks



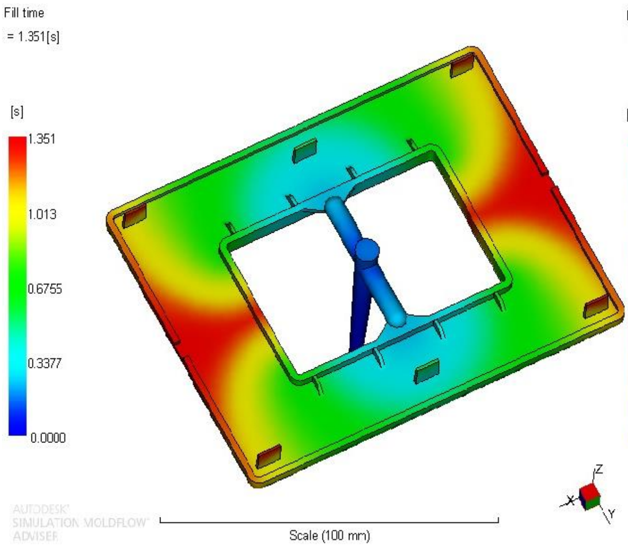
e) Air Traps



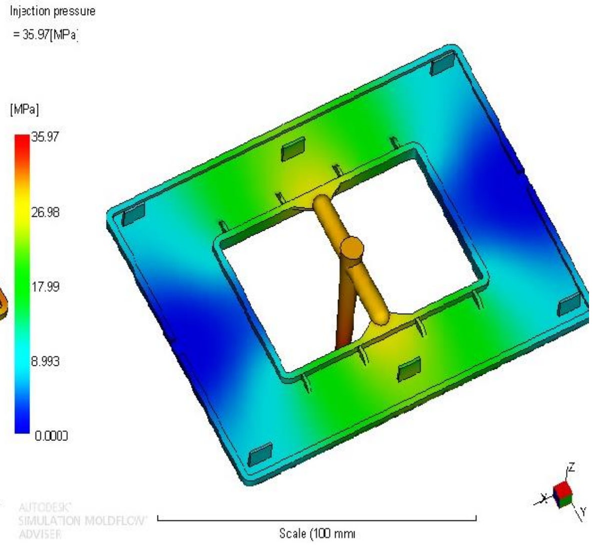
f) Weld Lines

Figure 5: Submarine Gate

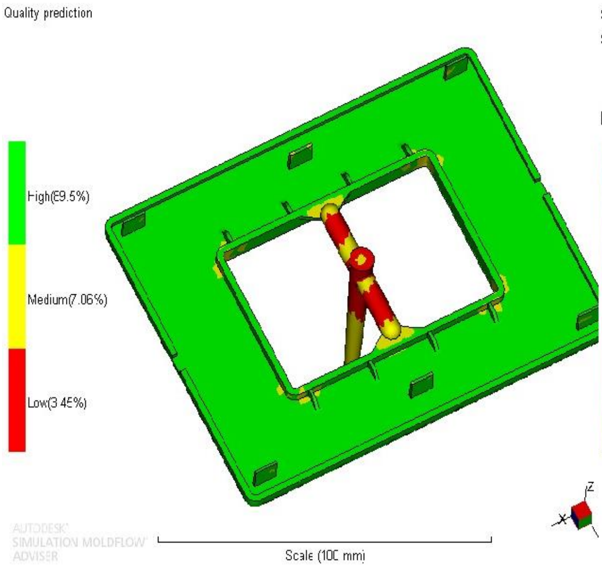
International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 7 Issue 03, March-2018



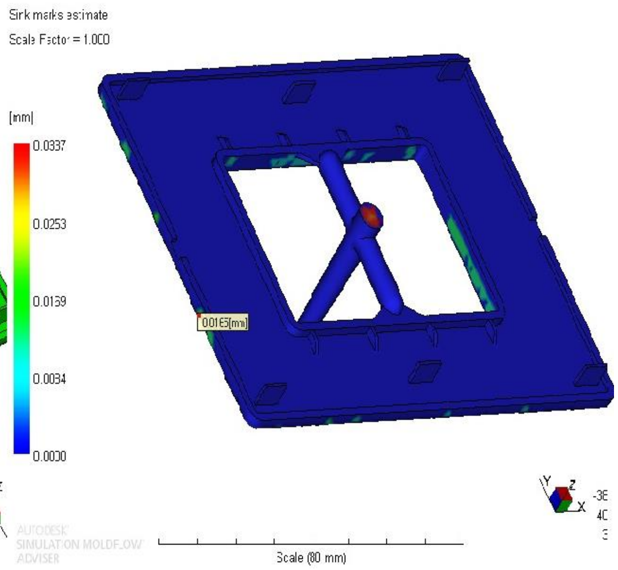
a) Fill Time



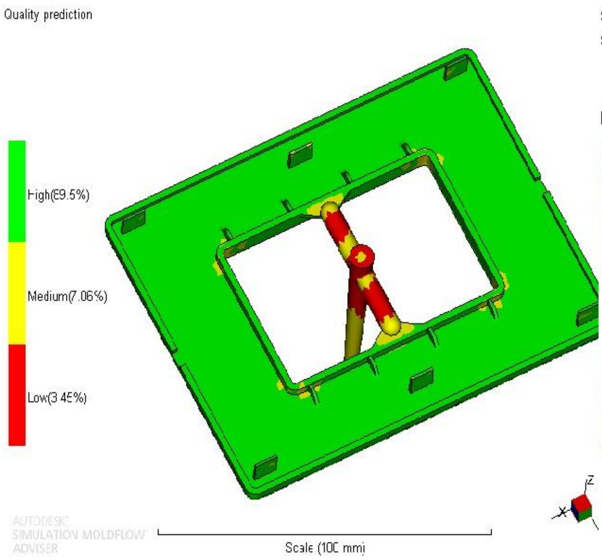
b) Injection Pressure



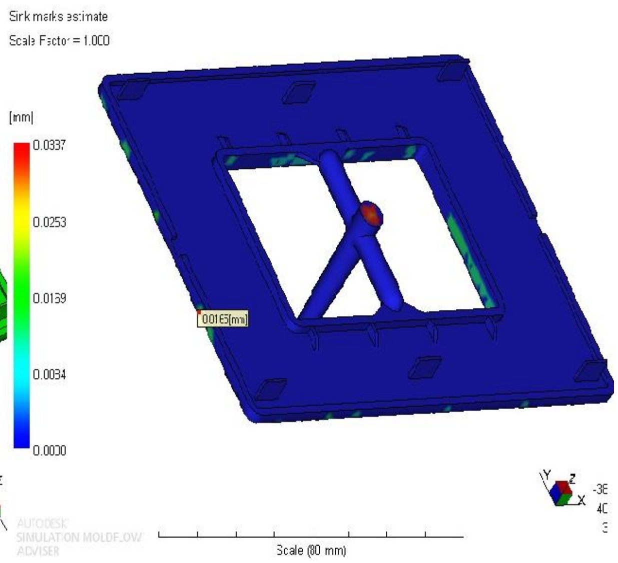
c) Quality



d) Sink Marks

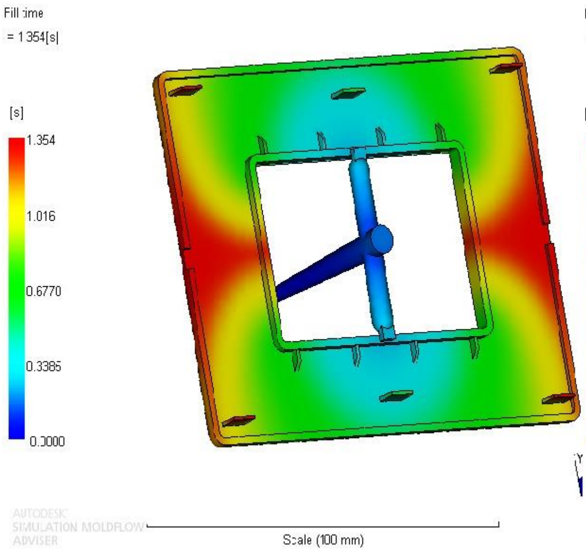


e) Air Traps

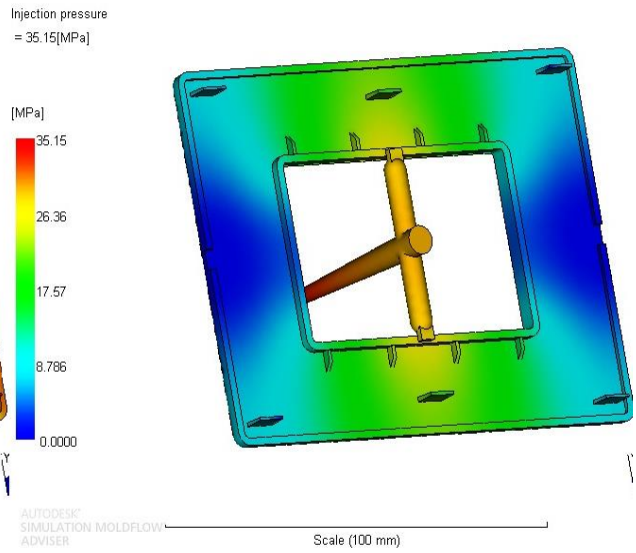


f) Weld Lines

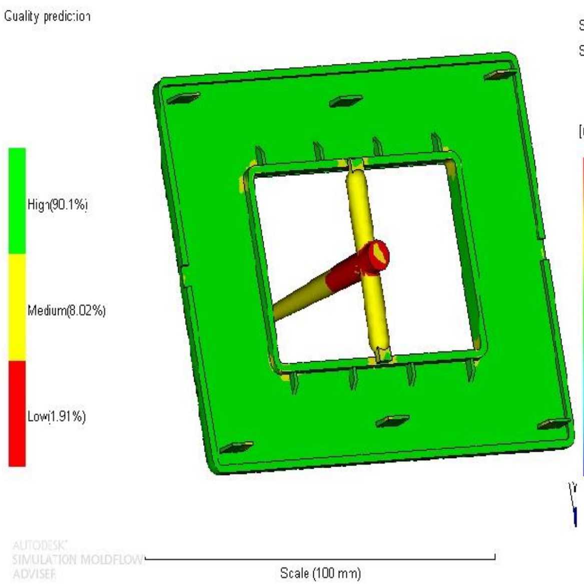
Figure 6: Fan Gate



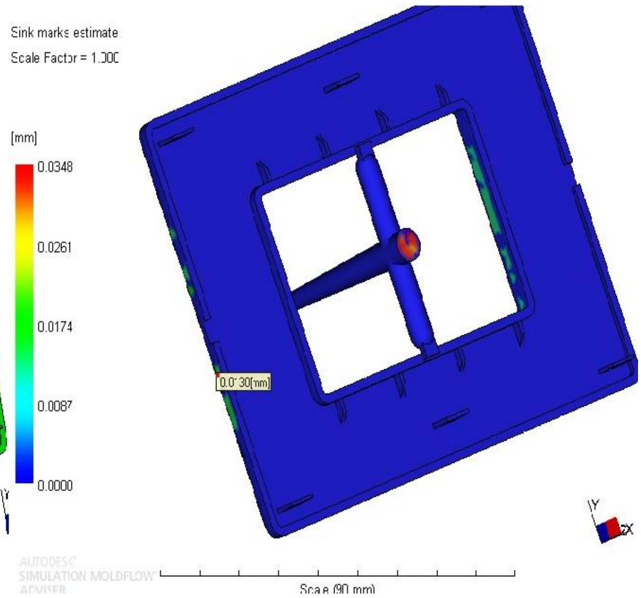
a) Fill Time



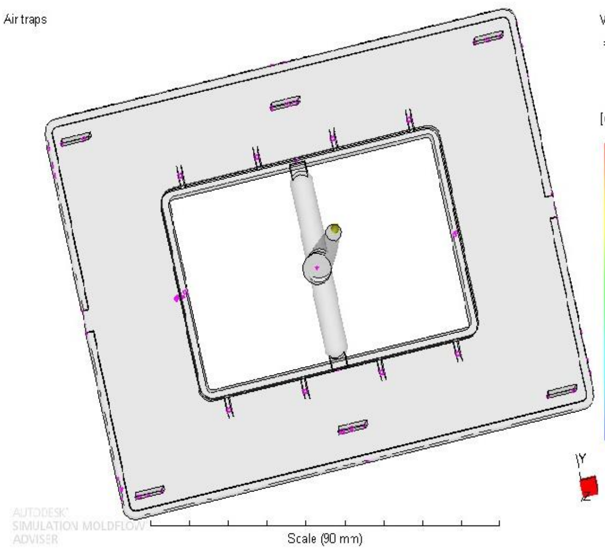
b) Injection Pressure



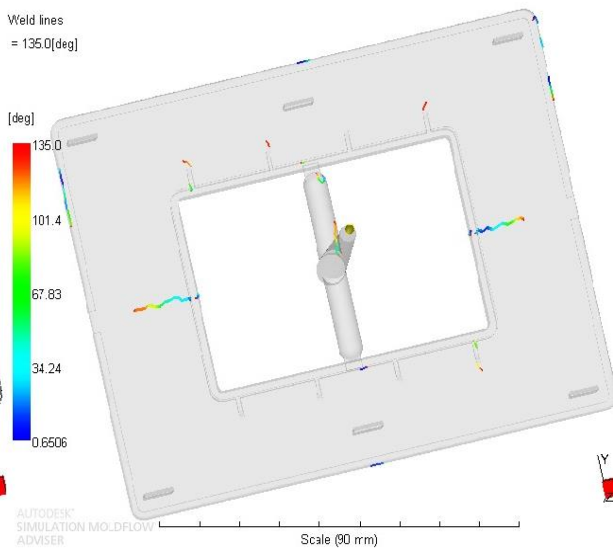
c) Quality



d) Sink Marks



e) Air Traps



f) Weld Lines

Figure 7: Overlap Gate

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 7 Issue 03, March-2018

VI.COOLING SYSTEM ANALYSIS

"w" wall thickness of the product mm (in)		"d" diameter of the cooling channels mm (in)		"a" center distance with respect to mold cavity	"b" center distances between cooling channels
2	(0.08)	8-10	(0.31-0.40)		
2-4	(0.08-0.16)	10-12	(0.40-0.47)	1.5-2d	2-3d
4-6	(0.16-0.24)	12-14	(0.47-0.55)		

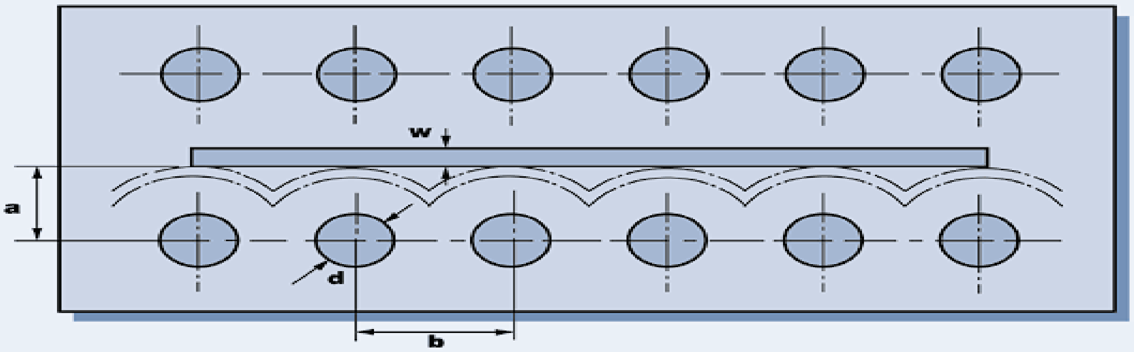


Figure 8: Cooling System Dimensions

Since, w = 2 mm for this product (see part details)
Therefore, d = 10 mm.
Let a = 25 mm.
For cooling analysis, certain parameters need to be specified:-
1. Diameter of cooling pipe, d = 10 mm.
2. Centre distance w.r.t. parting line = a + 5 mm = 2*d + 5 mm = 25 mm.
3. Coolant to be used = water.
4. Coolant flow rate = 1 lpm.
5. Inlet water temperature = 80° C.
6. Mould Material = P 20 Steel.
From Fundamentals of Plastic Mold Design by Sanjay .K. Nayak, we have,

$$Q = k*m_1*(T_{out} - T_{in}) \quad (1)$$

Where, Q = rate of heat extracted (kcal/hr)
k = constant to allow for heat transfer efficiency = 0.65
m_1 = mass of water passed (gm/hr)
$T_{out} - T_{in} = 5^oC$
Now, for this product, Q can be calculated as,

$$Q = M*[C_p*(T_1 - T_2)] + L \quad (2)$$

Where, M = mass of plastic material injected into mould per hour = 30 gm/shot× 60 shot/hr = 1800 gm/hr
C_p = specific heat of material (cal/gm/°C) =0.47
T_1 = injection temperature of material = 310°C
T_2 = mould temperature = 120°C
$Q = 1800\times[0.47\times(310 - 120) + 30]$
Q = 217476 cal/hr = 217.476 kcal/hr.
Substituting these values in (1), we get,
m_1 = 67 kg/hr = 1.133 lpm.
For cooling analysis, we have taken m_1 = 1 lpm (Refer Fig.9) so the coolant will do the desired job.

We have,

$$m_1 = (d*v*R)/3160 \quad (3)$$

where, m_1 = 1 lpm = 0.26 gpm
d = 10 mm = 0.4 inches
v = 0.35 centisokes at 80°C (Lanxess Design Guide)
R = Reynolds number
Substituting these values in (3), we get,
R = 5868.57 (Refer Fig.9)
It is always desirable to have Reynolds number in the range of 4000 to 10000 for effective cooling of the mould.
Therefore, the analysis is on correct path.

The moldflow simulation is done repeatedly for three different types of cooling systems and the one which gives best results for this product will be selected at the end. The parameters that will be looked upon to decide the best cooling system are:-
1. Temperature Variance
2. Frozen Fraction
3. Quality
4. Volumetric Shrinkage

1. Conventional Cooling System
It can be observed from Fig. 10 that, the maximum temperature variance is 25.73°C and maximum shrinkage is 8.792%. The product is of low quality but the frozen fraction in component is 7.614% although its maximum value for the entire component is 30.94% which will not be considered as maximum frozen fraction value as feeding system will be removed from the final product. So, now onwards frozen fraction value of final product will only be specified.

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 7 Issue 03, March-2018

2. Modified Conventional Cooling System

It can be observed from Fig. 11 that, the maximum temperature variance is 25.92ºC and maximum shrinkage is 8.811%. The frozen fraction in part is 7.571% but the product is of low quality.

3. Conformal Cooling System

In order to reduce the cycle time and control the uniform distribution of temperature, it is necessary to create conformal cooling channels which conform to the shape of the mold cavity and core. It can be observed from Fig. 12 that, the maximum temperature variance is 10.4ºC and maximum shrinkage is 8.783%. The frozen fraction in part is 7.27% also the product is of acceptable quality.

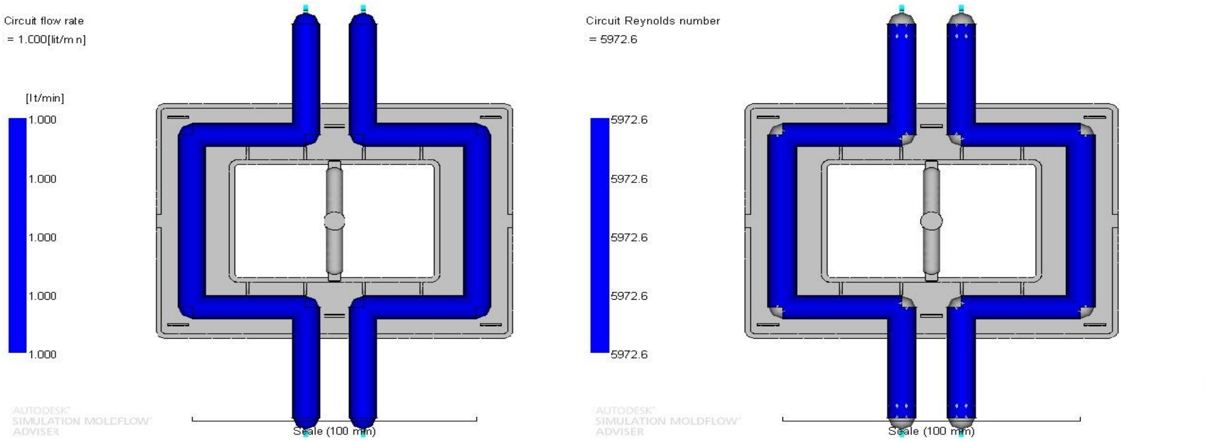
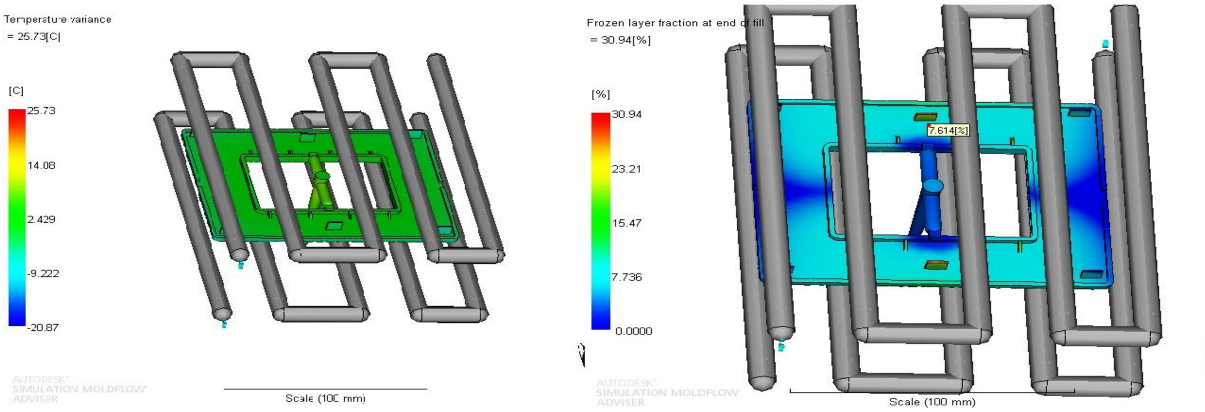
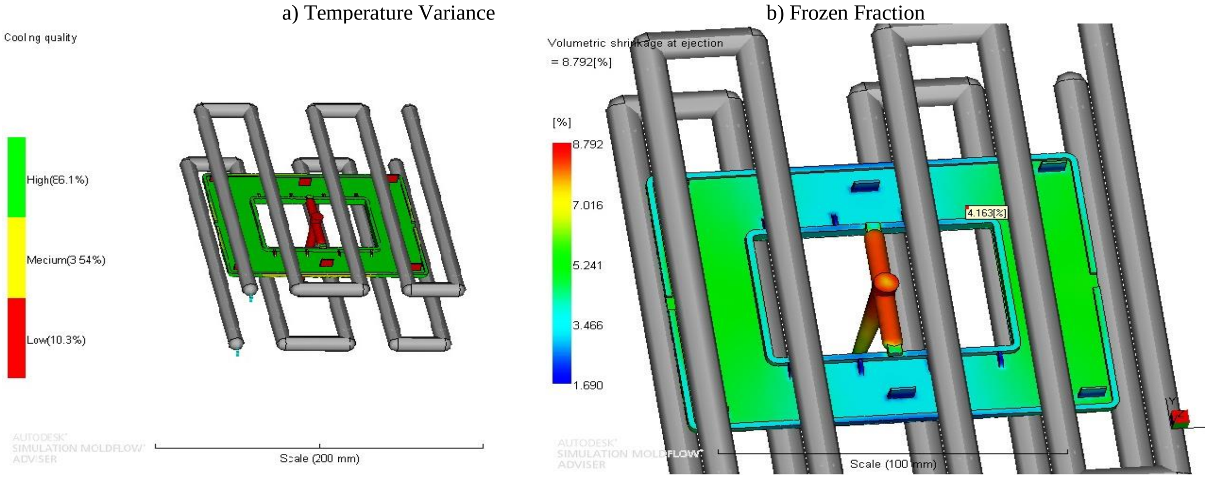


Figure 9: Coolant Flow



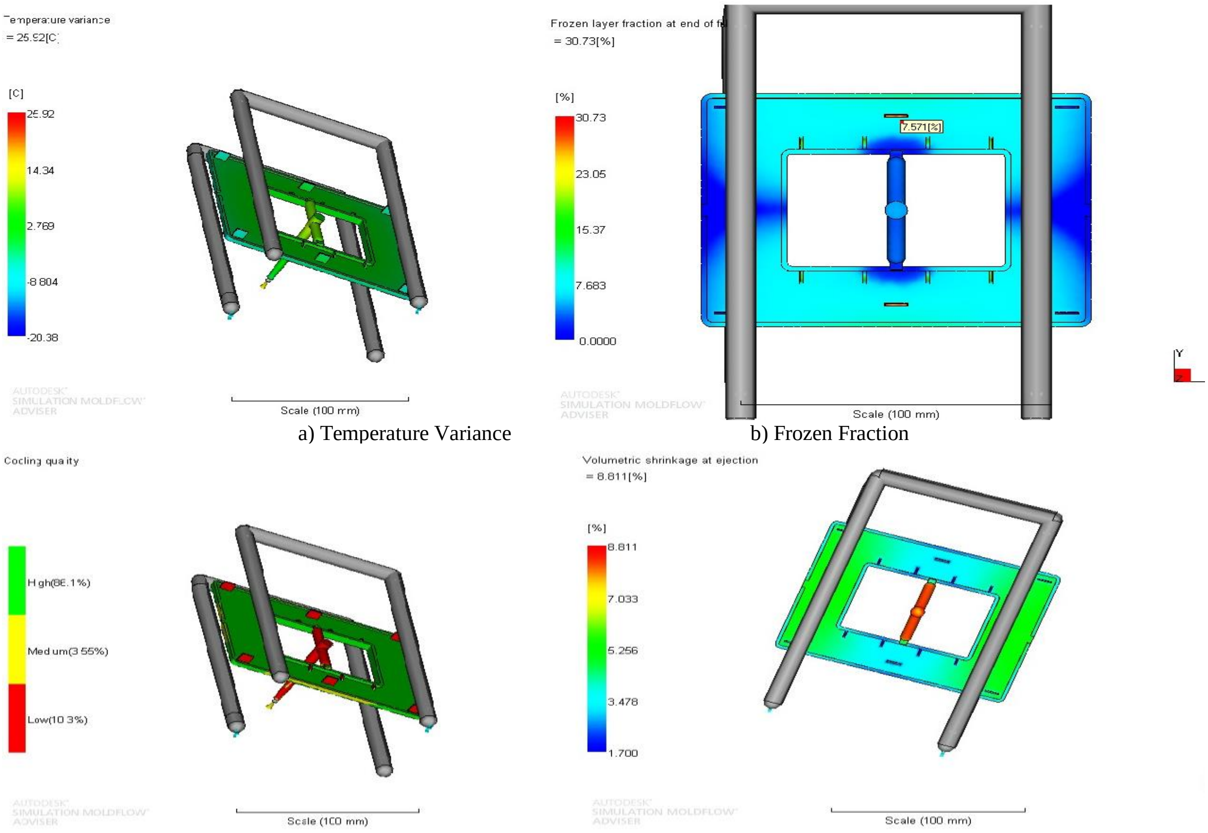
a) Temperature Variance b) Frozen Fraction



c) Quality d) Shrinkage

Figure 10: Conventional Cooling System

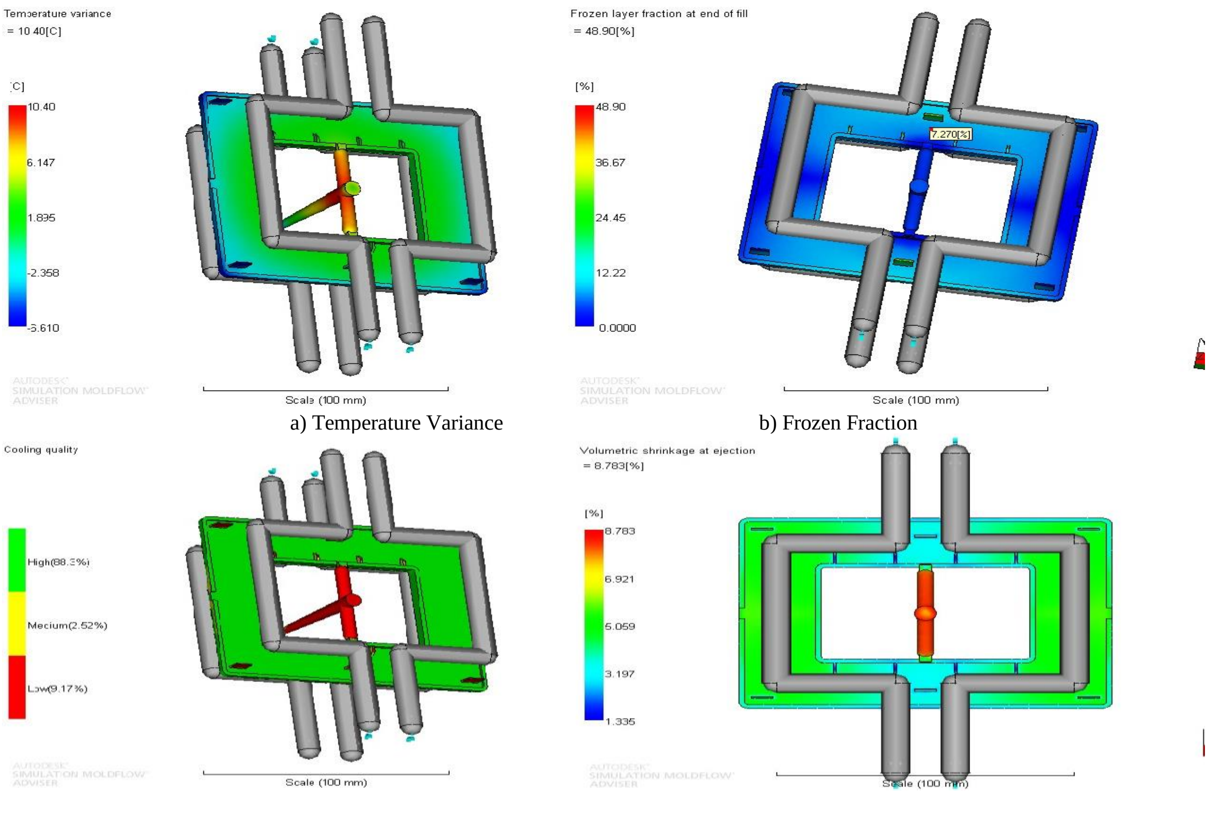
International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 7 Issue 03, March-2018



a) Temperature Variance b) Frozen Fraction

c) Quality d) Shrinkage

Figure 11: Modified Conventional Cooling System



a) Temperature Variance b) Frozen Fraction

c) Quality d) Shrinkage

Figure 12: Conformal Cooling System

Table 2: Types of Cooling Systems

Type of Cooling System	Conventional Cooling System	Modified Conventional Cooling System	Conformal Cooling System
Temperature Variance	25.73^{0}C	25.92^{0}C	10.4^{0}C
Frozen Fraction	7.614	7.571	7.27
Quality	Low	Low	Good
Volumetric Shrinkage	8.792	8.811%	8.783%

VII.CONCLUSIONS

In this paper, an optimum gate location is determined. There are five different trials done on type of gate to be used for this product. Out of these five trials, it is observed from Table 1 that Overlap Gate is optimum gate for this product. With use of overlap gate, the fill time is 1.354 s and it requires injection pressure of 35.15 MPa, also there are no issues of quality and degating.

There are three different types of cooling systems being analyzed. It is observed from Table 2 that Conformal Cooling System is the ideal cooling system for this product. It leads to better cooling properties due to exhibiting the lower volumetric shrinkage. It also provides the lowest time to reach the ejection temperature, which translates to lower cooling time and reduced overall cycle time. The conformal cooling channel shows near uniform cooling that makes it most favorable cooling system.

So, Use of an injection molding analysis software provides valuable information for plastic product and mold design in reducing time and cost of production especially for complex parts.

ACKNOWLEDGEMENT

Thanks to Prof. P.T. Mirchandani for his valuable contribution for guidance. It is my pleasure to acknowledge sense of gratitude to Prof. P.T. Mirchandani for their great support and encouragement in journal paper work.

REFERENCES

[1] Significance of Mold Filling Analysis for Finding Optimal Gate Location in Injection Molding Process for Bobbin. Mr. Yathish Kumar K.R*, Prof. Nagaraja.R**(International Journal of Scientific and Research Publications, Volume 4, Issue 4, April 2014 1 ISSN 2250-3153)

[2] A Simulation Study of Conformal Cooling Channels in lastic Injection Molding 1Shaileshbhai P Patel 2 Sudheendra S. 1Mechanical Design Engineering, 1Flora Institute of Technology, Pune, India(© 2017 IJEDR | Volume 5, Issue 1 | ISSN: 2321-9939)

[3] AUTHOR – Yathish Kumar K.R M.Tech. Student – Tool Engineering – pursuing, Government Tool Room & raining Centre, yathish.vinu@gmail.com

[4] LANXESS Corporation (A design guide for part and mold design)

[5] Injection Moulding Tool Design Manufacturing, Estimation and Comparison of L&T Power Box Side Panel Using Plastic Materials HDPE, ABS, PP and PC N. Sreenivasulu1 , Dr. D. Ravikanth2 1 (M. Tech Student, Dept of Mechanical Engineering,K.S.R.M College of Engg, Kadapa, Andhra Pradesh, India) 2 (Associate Professor, Dept of Mechanical Engineering, K.S.R.M College of Engg, Kadapa, Andhra Pradesh, India) [IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE) e-ISSN: 2278-1684,p-ISSN: 2320-334X, Volume 8, Issue 3 (Sep. - Oct. 2013), PP 23-32]

[6] CONCEPTUAL DESIGN OF TWO PLATE INJECTION MOULD TOOL FOR FIVE PIN DAIMLER REGULATOR Kiran Tom Thomas1* and Ramesh Babu K1(ISSN 2278 – 0149 www.ijmerr.com Vol. 3, No. 4, October 2014 © 2014 IJMERR.)

[7] Knowledge.Autodesk.Com/Support/Moldflow-Adviser