ISSN No:-2456-2165

Mould Flow Analysis of a Plastic Injection Moulding Component Using Software Tool to Obtain a Quality Output

Sowrabh Kugunavar*
M.Tech in Tool Engineering.
GT&TC- Mysore, India.

L.G. Sannamani
Dept of PG studies
GT&TC- Mysore, India.

Abstract:- Plastic injection molding is a versatile process in which most of complex parts for various application can be produced. Growth in technology has made favourable software tool for the manufacturers to make best use of them in the field which a quality-based approach can be made even before the mould tools are manufactured. One such software tool used in this article is moldflow. The prime objective of this article is to ensure that the defects which causes the deterioration of the quality of the product to be produced are eliminated and this is done by using analysis software tool to study and interpret the plastic flow behaviour in the cavity. Various parameters like filling, packing, pressure, temperature etc… which play a major role towards the moulding process are taken into consideration.

Keywords:- *Injection Molding; Quality; Software Tool; Flow Analysis; Moldflow; Solidworks.*

I. INTRODUCTION

With the growing technology of tooling, it is witnessed that a lot of advancement in the field is being done. Injection molding is one of the tooling processes that was developed based on the principle of pressure diecasting process by JW Hyatt and IS Hyatt. They patented their stuffing machine in USA in 1872 [1]. This technique evolved into modern day plastic injection molding machines. Plastic moulding process involves 4 stages in a cycle, mould closing or clamping, injecting the plastic, cooling of the mould and component, ejection of the component and closing of the mould. By this process one can produce a highly accurate with satisfying aesthetics and quality products at lower lead manufacturing time. Plastic injection molding takes up to 32% among various plastic production technologies [2]. The quality of the product can be defined as its aesthetics, mechanical strength or its appearance. Also, it is classified under four major categories that are processing conditions, moulding machine performance, material characteristics and mould tool design [3]. In this article the quality is focused, based on the analysis performed on the component so that the study of the results obtained assists the mould tool to be designed appropriately.

To perform the analysis computer aided tool called Moldflow is used. This software tool facilitates the user to analyse the behaviour of the plastic to be moulded when it is injected into the cavity of the mould tool. With various parameters like fill, pack, shrinkage, pressure, temperature etc… obtained, the user can infer the substantial results to improve the quality of the product.

II. MATERIAL AND COMPONENT DETAILS

The material of the product recommended by the customer based on its application is NYLON 66, 33% glass fibre filled polymer. The melt density of the material is 1.15 g/cm3 and the solid density is 1.36 g/cm3. The recommended melt temperature range by the software is 290°C to 305°C.The recommended mold temperature by software is around 65°C. The material structure is crystalline.

The component is modelled as per customer requirement and the drawing details with the help of solidworks design software tool. The metal inserts were excluded to perform the analysis and it is done because the software is recommended only for plastic flow analysis. The computer aided design (CAD) model of the component is as shown in the *Fig. 1.*

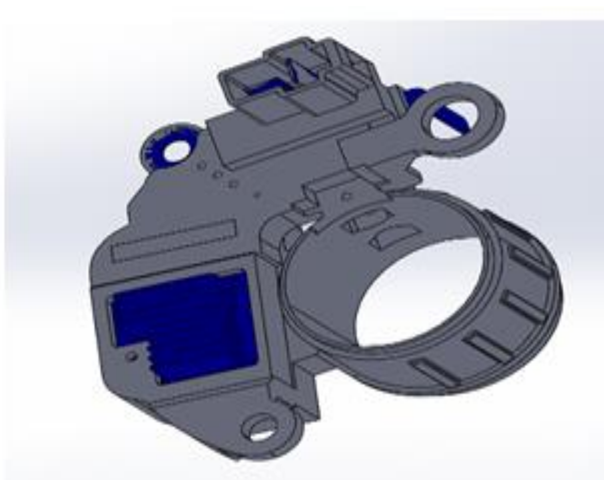

Fig 1:- Component CAD model

III. METHODOLOGY

- The following steps are followed in order to perform the analysis for gating suitability is as shown in the flow chart *Fig. 2*. This is the primary analysis.

- The next step is to design the feed system with the help of CAD software.

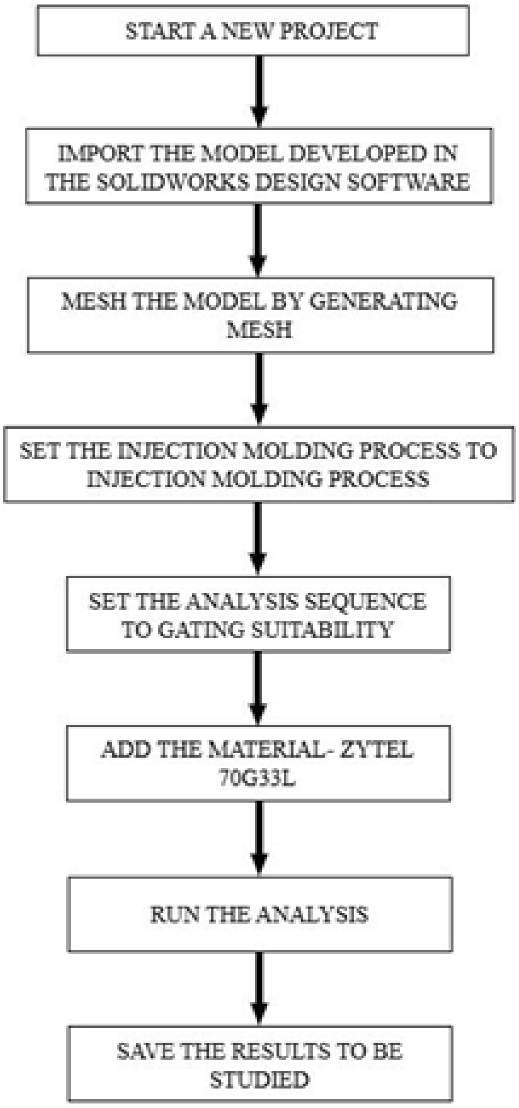

Fig 2:- Procedure to analyse gating suitability

- The following steps are followed in order to perform the analysis for filling, packing and shrinkage which is as shown in the flow chart *Fig. 3*. This is the secondary analysis.

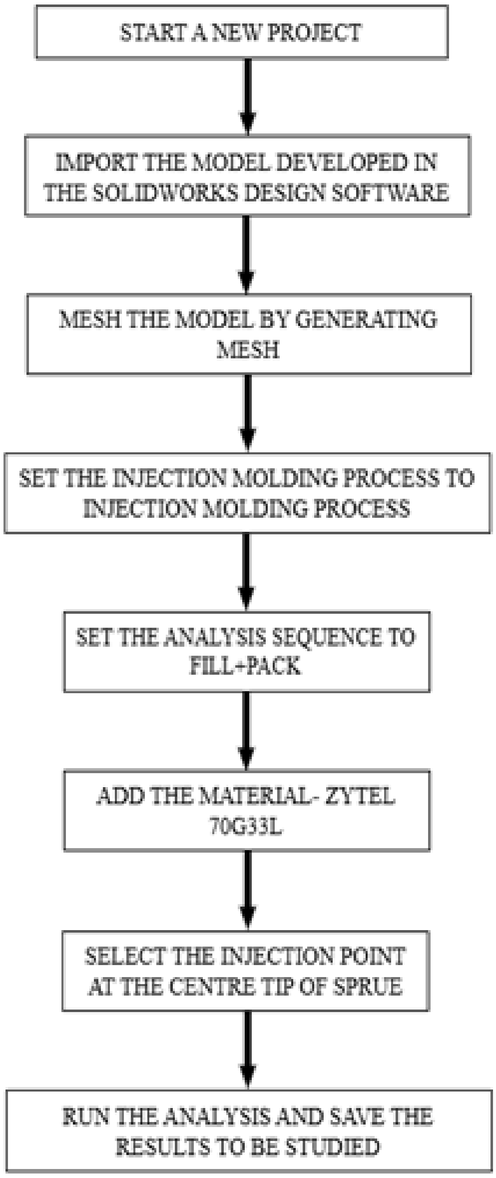

Fig 3:- Procedure for fill, pack and shrinkage analysis

IV. DESIGN OF FEED SYSTEM AND ANALYSIS

A. Primary Analysis

Upon running the gating suitability analysis, the results obtained are to be studied in order to design the feed system. The following are the graphic results obtained.

- *Fig. 4* shows the best suitable region for the gate from blue (indicating best region) to red (indicating worst region). On investigation the cylindrical region in blue is preferred for injection of plastic because the region does not interfere with the metal inserts that are placed and also it is favourable for submarine type of gate.

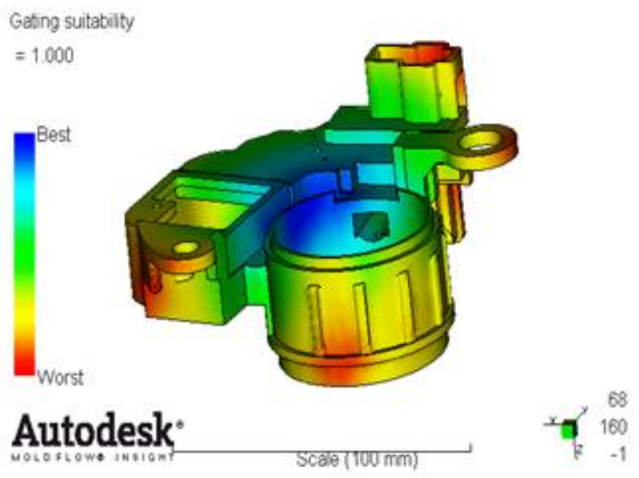

Fig 4:- Suitable position for the injection gate

- Upon comparison study of *Fig. 4* and *Fig. 5* help to understand the flow resistance of the injected material is lowest at the inner area of the cylindrical portion therefore the region preferred for the gate position is favourable.

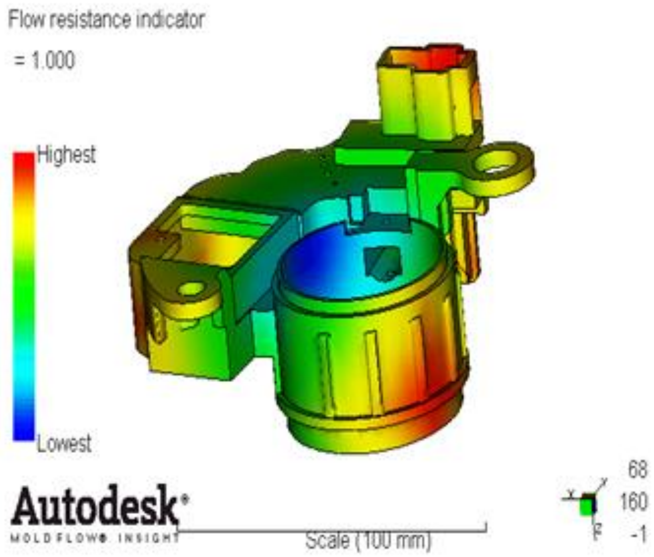

Fig 5:- Indication of flow resistance of the material to be injected in the cavity

- The *Fig. 6* indicates the gate location by moldflow adviser. The gate is the yellow conical structure. Thus, by inferring the data obtained, the design of the feed system is to be done.

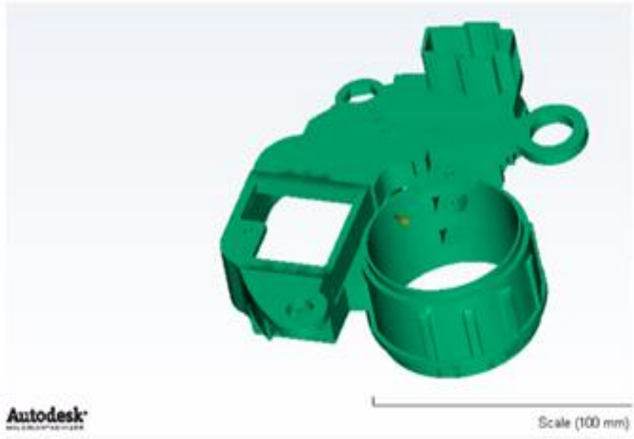

Fig 6:- location of the gate

V. DESIGN OF FEED SYSTEM

In plastic injection molding the feed system refers to the flow path of the material from sprue to the cavity entry. The feed system consists of sprue, runner and gate. The submarine gate is designed at a subsurface level at an angle of 45°. The length of the runner (L) is 15.5mm and the weight of the molding (Ws) is 57.22 grams. The depth of the runner

(Rd)= (√(Ws)×√(4L))/(3.7) is the formula used [4]. On substituting the values of Ws and L, the depth of the runner Rd= 3.96mm. a value of 5mm is taken for the ease of manufacturability. The sprue length is taken as per the depth of the mould. The larger diameter of the submarine gate is taken to be 2/3rd of the depth and the smaller diameter at the entry of the cavity is taken to be 0.5mm [5]. The dimensions of the runner are as shown in the *Fig. 7*.

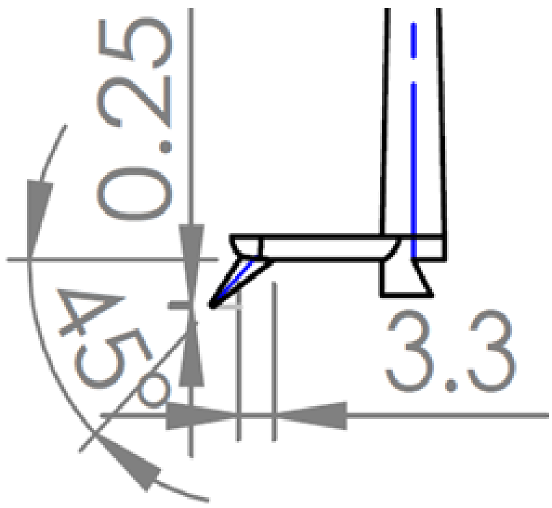

Fig 7:- dimension for feed system

VI. SECONDARY ANALYSIS

The results obtained by the secondary analysis are as follows.

- The *Fig 8* indicates that the time required to fill the cavity is 2.294 seconds. The filling of the material after the mould tool closes. The blue region indicates the fill at initial time and the region in red fills at the end or final time where the material in the cavity reach extreme corners.

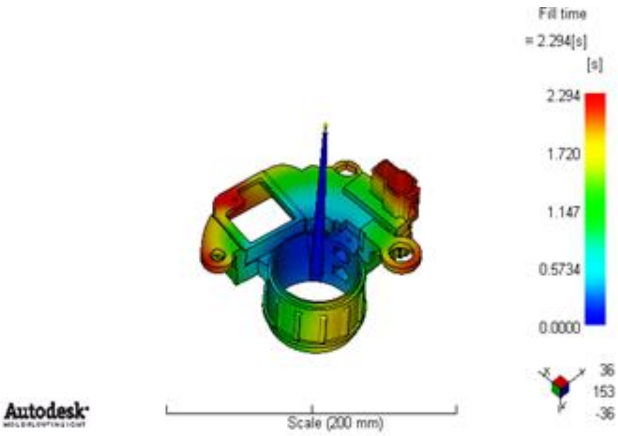

Fig 8:- Fill time of the material inside the cavity

- *The* temperature at the flow front is a result by fill analysis, this indicates that the polymer temperature when it reaches a specific point. Referring *Fig. 9*, it can be observed that around 60% of the area has the highest temperature at the flow front and to overcome this the injection pressure has to be varied [6].

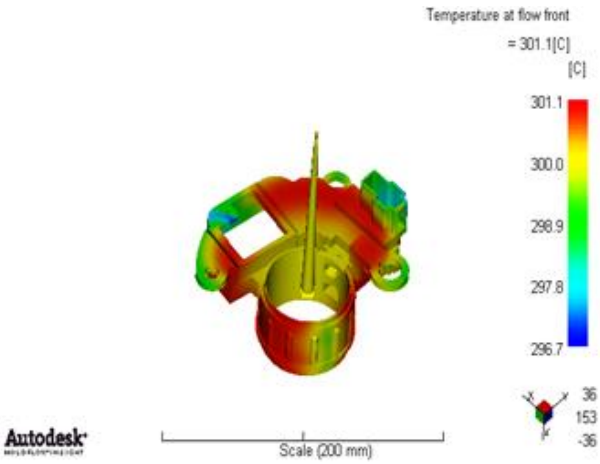

Fig 9:- Temperature of the polymer at the flow front

- *Fig. 10* shows the maximum value of the Injection pressure that is obtained during the filling phase prior to the velocity/pressure switchover. Initially the pressure is said to be at 1 atmosphere throughout the mould. The value of the pressure is highly dependent on the resistance offered by the material in the mould thereby viscosity plays a major role. From the *Fig. 10* it is observed that the pressure distribution is not varied largely, also in practicality ±10bar is the allowance is considered [7].

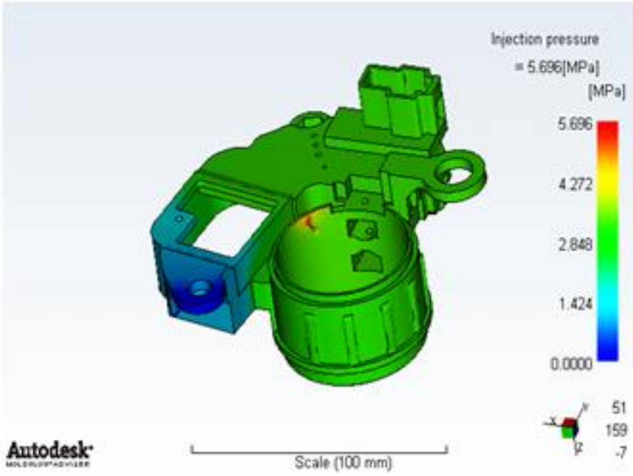

Fig 10:- Injection pressure required to overcome the resistance offered by the polymer

- The clamping force is a time series versus tonnage XY plot. It is applied in order to held the core half and the cavity half of the mould in position during the injection of the polymer. Referring the fill time of 2.294 seconds, from the graph *Fig. 11* it is observed that the clamping force required is 1.09 tonne. In practicality only 80% of the machine efficiency is considered for the safety purpose [8].

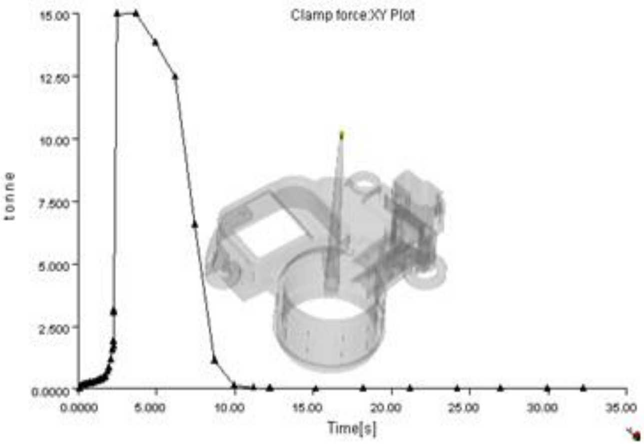

Fig 11:- Tonnage v/s time plot inferring the clamping force

- Sink mark is one of the defects occurring in the molding process. It always occurs on the surface of the component either due to lower injection pressure or lower packing pressure. Shorter cooling time is also one of the reasons for sink marks to appear. From *Fig. 12* it is seen that there are absolutely no sink marks appearing, it is a mere indication that the shrinkage is even in the part. This directly increases the quality of the component.

ISSN No:-2456-2165

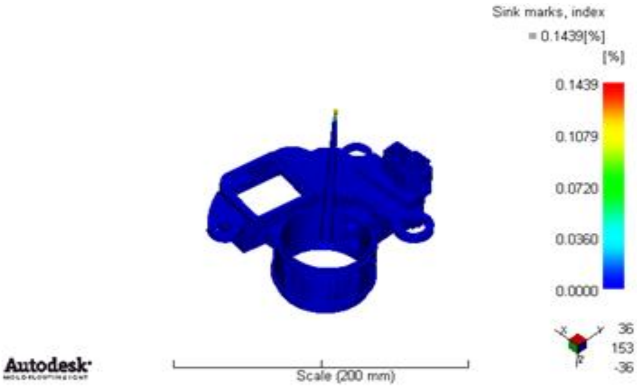

Fig 12:- Sink mark index study

- Weld lines are the defects in injection molding. They are the lines that occur when the molten plastic flows around the recess or holes and knit at a point. These are one of the major defects to decrease the strength of the component. Although it is unavoidable, its effect on strength can be reduced by changing gate location. *Fig. 13* shows the best gate location for which the weld lines are very minimum.

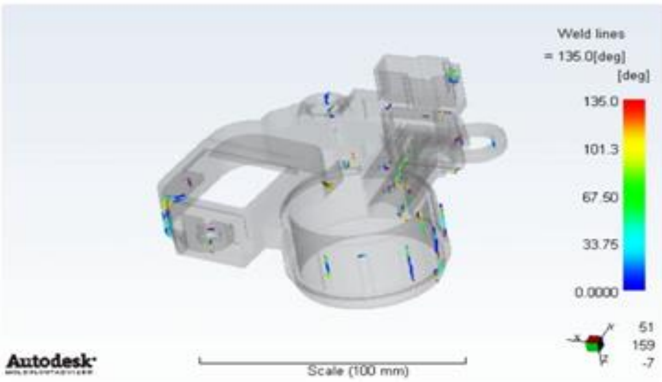

Fig 13:- Detailed view of weld lines

- Air traps in injection molding occur due to air bubbles that are trapped when the flow front of the polymer coincides. This defect causes incomplete filling, blemish and packing. From analysis the detection of the air trap possibilities is indicated in the pink bubbles as shown in the *Fig 14*. This defect will reduce the quality of the component, hence to avoid it proper air vents has to be provided in the mould tool.

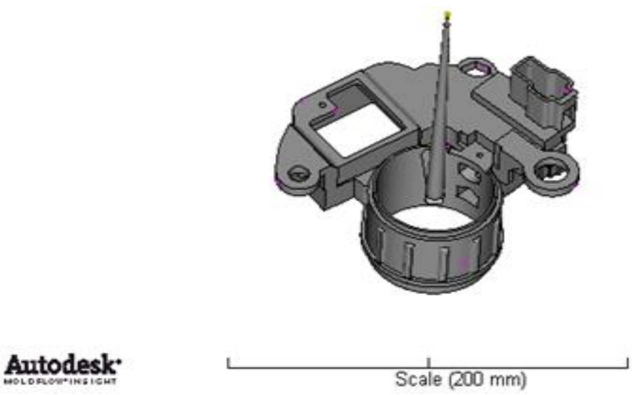

Fig 14:- Air trap indication with pink coloured bubbles

VII. CONCLUSION

The above study made on various parameters, are done to ensure the quality of the component is maintained. The reliability on the software is higher when compared to manual calculations and analysis. The methodology followed in this article is valid to all the plastic components that are to be injection moulded. Even before a mould tool is manufactured a quality-based approach to develop the mould tool, eliminates the errors caused by the defects as well as help the designer with the right processing conditions.

REFERENCES

[1]. P. Jones, "The Mould Design Guide."

[2]. P. Shakkarwal, "Design and Mould Flow Analysis of Injection Mould for Connecting Link," International Journal on Emerging Technologies, vol. 4, no. 1, pp. 182–185, 2013.

[3]. B. H. Min, "A study on quality monitoring of injection-molded parts."

[4]. R. G. W. PYE, INJECTION MOULD DESIGN, Fourth. .

[5]. A. U. Holla MTech Student and R. N. Senior Lecturer, "Determination of Process Settings for Injection Moulding Tool by use of CAE Software Tools."

[6]. "TEMPERATURE AT FLOW FRONT." [Online]. Available: https://knowledge.autodesk.com/search-result/caas/CloudHelp/cloudhelp/2015/ENU/MoldflowInsight360/files/GUID-2BC51F30-583F-4E5C-8824-1BF7BAF0A144-htm.html.

[7]. "INJECTION PRESSURE." [Online]. Available: https://knowledge.autodesk.com/support/moldflow-adviser/learn-explore/caas/CloudHelp/cloudhelp/2015/ENU/MoldflowAdvisor/files/GUID-2FE30034-2A02-47CE-823B-694E3BD81191-htm.html.

[8]. "CLAMPING FOCE." [Online]. Available: https://knowledge.autodesk.com/support/moldflow-insight/learn-explore/caas/CloudHelp/cloudhelp/2017/ENU/MoldflowInsight/files/GUID-76E9C735-4C1F-4AF9-A54E-B7646D9A9198-htm.html.