International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 4 Issue 05, May-2015

Determination of Process Settings for Injection Moulding Tool by use of CAE Software Tools

Ajith Holla U
M.Tech Student
Department of PG Studies,
Govt. Tool Roomand Training Centre,
Mysore.

Ramesha N
Senior Lecturer
Department of PG Studies,
Govt. Tool Room and Training Centre,
Mysore.

***Abstract*—This paper deals with various aspects involved in designing an Injection Moulding Tool for a Thermoplastic component. For good quality of the component, the design of the Injection Moulding Tool has to be good. This can be achieved by optimal selection of process settings. To facilitate this, software tools can be used. Usage of software to ascertain process settings, which is used in designing can be called Computer Aided Engineering (CAE). CAE tools provide means to validate the design of Injection Moulding Tool and help to ascertain valuable simulation data which can be used to further improve the design. It also reduces time and saves material, which otherwise would have been wasted during testing and trial. MoldFlow was used to ascertain suitable gate position, and to get valuable data regarding temperature, pressure and flow patterns during the injection process.**

***IndexTerms*—Injection Moulding, CAE, Software Tools, MoldFlow, Flow Analysis.**

I. Introduction

Injection Moulding is one of the versatile manufacturing methods for plastic components. It involves melting the plastic material and injecting it into the Mould under pressure and allowing it to cool and solidify inside the Mould cavity to form the shape of cavity. It is relatively recent invention. The Injection Moulding was developed based on the concept of Pressure Die Casting. J S Hyatt and J W Hyatt patented a ‘stuffing machine’ in 1872, which became predecessor for the modern Injection Moulding Machine [1]. Then the Mould is opened and the component is ejected. This process can be repeated many times and each time the component has same dimensions. Injection Moulding process allows producing wide range of plastic products which varies in shape and size and they can be manufactured to high tolerances [2].

For high quality component design of the Mould has to be good [3]. So, the design should be done considering the material of the plastic part, its liquid state properties etc. This can be done easily using software tools like MoldFlow, which allows to analyze how the plastic flows and behaves inside the Mould once injected. MoldFlow also helps to identify Injection points for smoother fill and packing of material [4].

Temperature and Injection Pressure are the two most important parameter for quality of component [5]. Analysis has been done to find out the suitable gating position and to determine melt temperature to get optimal fill time, cooling time and quality of product. This analysis has also helped in identifying regions where air traps might occur. This allows for optimizing of the whole Injection Moulding Process without having to perform trial runs, thus saving time and money.

II. Material and Methodology

Material of the component is Polypropylene (PP). Weight of the component is 2.1 g. Values of Process Parameters has been recommended by the producer as follows: Melt Temperature is 240 ºC. Ejection Temperature is 125 ºC. Mould Temperature is recommended to be around 60 ºC. Maximum Injection Pressure is selected as160MPa. Data were obtained from MoldFlow Adviser Database for the Polypropylene. Velocity – Pressure switchover is assumed to be 99% of volume.

The process parameters mentioned above were utilized during analysis. Simulations were done to identify suitable gate position and using that runner system and suitable gating was selected. Then simulation was done for fill time, temperature of the melt during flow and pressure drop across the cavity of Mould. Moulding Window Analysis was performed to ascertain the feasibility of the selected process settings.

III. Runner and Gate

A. The*Gate Position*

The location of the gate on the molded part plays a major role in how the part will perform, as well as the quality, properties, and performance of the part. Gate position is one of the key aspects for better quality product [6]. So simulation was done to determine flow resistance and hence determine suitable gate position. Resistance to flow was ascertained by the simulation to determine the least resistant point.

Figure 1 shows that the resistance to injected material is less at the side and at the top surface. Red indicates High resistance. There is more resistance at the chunky rear end and the thin sections. Figure 2 indicates suitability of injection point based on resistance to flow.

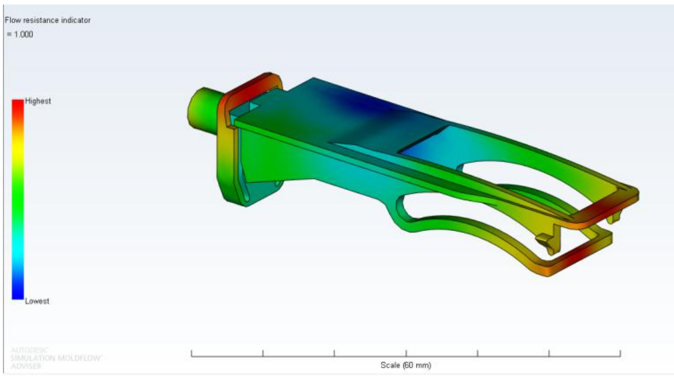

Fig.1 Resistance to Flow

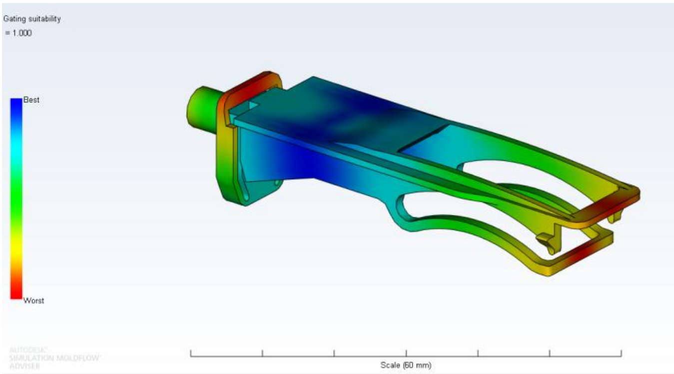

Fig.2 Gating Suitability

From both the figures it is clear that top surface and side surface is suitable for injection. Since the top surface needs to be smooth without any residual marks.But, the top surface is thinner, which is not suitable for gate location [6]. Hence, the side of the component is selected for Gate location as seen in Fig 3.

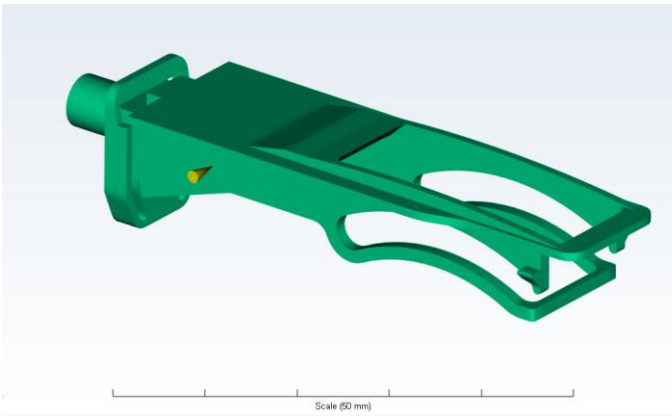

Fig.3 Gate position selected

B. *Runner and Gate Design*

Complexity of the component, which requires 3 side cores, dictates use of two cavity mould. This also helps to maintain runner balance [7]. Hence, the runner layout is selected as shown in Fig. 4. Length of runner comes to 142 mm. Also, weight of moulding comes to 4.2 g.

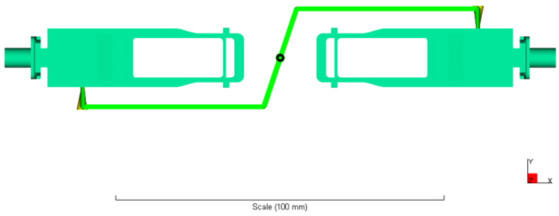

Fig. 4 Runner Layout

Runner dimensions was calculated using the following formula [7]:

$$\text{Runner depth, } D = \left(\sqrt{w} \times \sqrt[4]{L}\right) / 3.7$$

where, w = weight of moulding = 4.2 g
L = Length of Runner = 142 mm

From the above equation, we get runner depth as 1.91 mm which is increased to 2 mm for ease of manufacturing. The shape of runner is modified trapezoidal, as shown in Fig. 5. Since the gate location is below the parting surface, submarine gate is selected, as shown in Fig. 6. For Submarine gate, larger diameter is taken as 2/3rd of runner depth and smaller diameter as 0.5 mm.

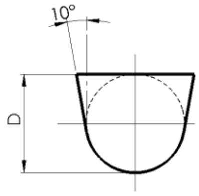

Fig. 5 Modified Trapezoidal Gate

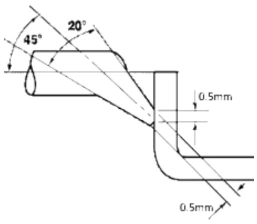

Fig. 6 Submarine Gate

IV. Flow Analysis

Simulation was done to determine Fill time, Temperature at flow front, and pressure drop across the mould cavity. This was compared against results of Molding Window Analysis. Molding Window Analysis gives suitability of Process parameters like Melt Temperature, Mould Temperature and Fill time. The results are in a chart of color gradient from green to yellow to red, with green being preferred. Molding Window chart is shown in Fig. 7.

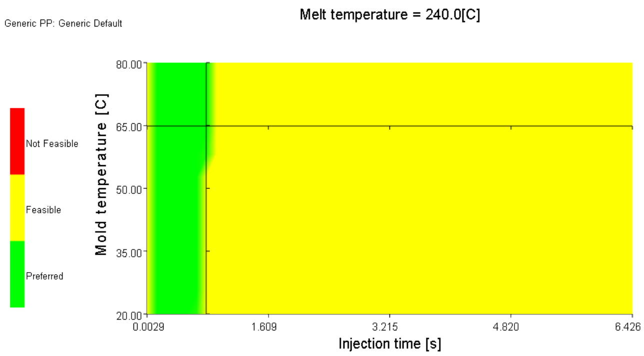

Fig. 7 Molding Window

Fig. 8 – 11 shows the results of flow simulation. Those are used to ascertain the feasibility of the parameters selected compared with Molding Window Analysis.

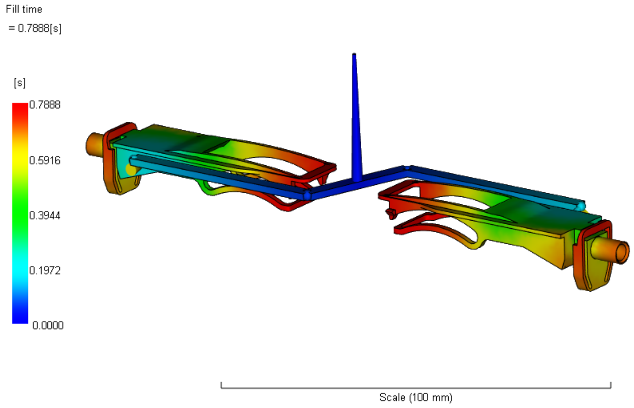

Fig. 8 Fill Time

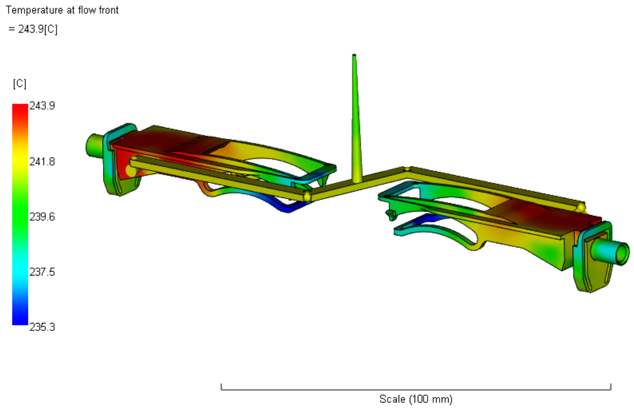

Fig. 9 Temperature at flow front

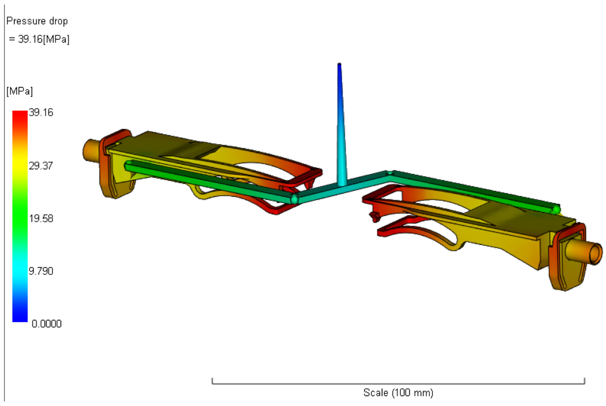

Fig. 10 Pressure drop across cavity

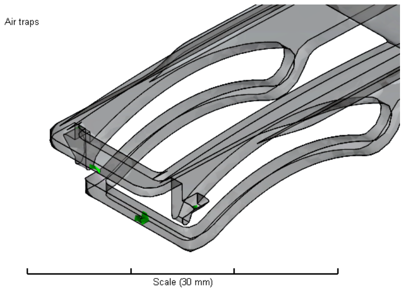

Fig. 11 Air Traps

Comparing Fig. 7 and Fig. 8, it seen that the fill time of 0.7888s is acceptable for Melt Temperature of 240 ºC and Mould Temperature of 65 ºC. From Fig. 9 it is seen that the minimum flow front temperature is 235 ºC, which is well above the melting temperature of PP. This indicates that there is little possibility of cold shut due to temperature factors.

From Fig. 10, it is seen that maximum pressure drop is 39.16 MPa. Pressure drop indicates minimum pressure required to push the material across the mould cavity. So, a minimum pressure of 40 MPa is sufficient to push the material across whole of the mould cavity. But, for compact filling at least 4 times the pressure drop is suggested as the injection pressure. Hence, Injection Pressure of 160 MPa is selected.

From Fig. 11, air trap seems to occur at the narrow portions. These can be eliminatedby providing venting on core and cavity inserts [7].

V. CONCLUSIONS

The paper shows the analysis result from the simulation using MoldFlow Adviser Software. Using MoldFlow optimum position of Gate position is determined. Using that, Gate and runner were designed. Runner depth is 2 mm and runner length comes to 142 mm. Gate selected is submarine gate.

Simulation results for Fill time and Molding Window Analysis indicated that Melt temperature of 240 ºC, Mould Temperature of 65 ºC and Fill time of 0.7888 s resulted in compact filling. Also, Injection Pressure of 160 MPa was deemed sufficient to completely fill the mould cavity at the given temperatures. Minimum Flow front temperature was at 235 ºC, which is sufficient to eliminate cold shut problems.

Simulation helped to improve the quality of product and reduce cost by shortening design time.

ACKNOWLEDGMENT

This work has been completed with support and guidance from Sandeep Kumar, Jai Shakthi Tools, Bangalore and N Ramesha, Senior Lecturer, Dept. of PG Studies, Govt. Tool Room and Training Centre, Mysore.

REFERENCES

[1] Peter Jones "The Mould Design Guide", ISBN: 978-1-84735-087-9

[2] Dominick V. Rosato, Donald V. Rosato, Marlene G. Rosato "Injection Moulding Handbook", 3rd Edition, ISBN 0-7923-8619-1

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

[4] http://www.autodesk.com/products/MoldFlow/overview

[5] OpranConstantin Gheorghe, Teodorescu Draghicescu Florin, Gheorghe Tudor Vlad and DobrescuTiberiu Gabriel "Optimization of Micro Injection Moulding of Polymeric Medical Devices using Software Tools", 24th DAAAM International Symposium on Intelligent Manufacturing and Automation, 2013, Procedia Engineering 69 (2014) 340 – 346.

[6] Charles A Harper, "Handbook of Plastic Processes", ISBN: 978-0-471-66255-6

[7] R G W Pye, "Injection Mould Design", Fourth Edition, ISBN: 81-7671-010-5