International Journal of Research Publication and Reviews Vol (2) Issue (8) (2021) Page 721-727

International Journal of Research Publication and Reviews

Journal homepage: www.ijrpr.com ISSN 2582-7421

Flow Analysis for an Injection Mould Tool for the Compnent ION Base Plate

***Prekshith Y. P.*[1] ** Dr. S.N.Ravi Shankar.*[2] ** L. G. Sannamani*[3]**

Student, M.tech in tool engineering, GT&TC, Mysuru-16, INDIA
Principal, Post Graduate studies, GT&TC, Mysuru-16, INDIA
Head of the department, Post Graduate studies, GT&TC, Mysuru-16, INDIA

ABSTRACT

The study gives an insight on analysis of base plate manufactured by injection moulding process. The component used for this study is ION base plate used in the assembly of an alpha ventilator. This study is done to understand the various parameters affecting the moulding process. Analysis is done by using the software Autodesk Moldflow adviser, by which various parameters such as, best gate location confidence of fill, weld lines, warpage etc., are analyzed. By using the results obtained by the analysis the mould optimum design of the mould is possible. The information presented in this paper may be especially beneficial during the early stages of mould/part design, when any changes can be made easily and cost-effectively.

Introduction

Thermoplastics injection moulding is recognized as one of the most significant procedures for producing plastic products.It starts with solid plastic material being fed into the hot injection barrel through the hopper. The injection screw revolves during the plastication step, transporting molten material to the screw chamber in front of the screw tip. Plasticization comes to a halt when a sufficient amount of molten material has been prepared. The part to be created is achieved during the filling stage by introducing molten material into a mould cavity. The injection gate's position is critical because it can affect the flow direction and melt solidification during and after filling.

The packing stage follows the injection stage when the cavity is nearly full, during which extra pressure is applied to force more molten material into the cavity to compensate for material shrinkage. The cooling stage next eliminates any remaining heat from the melt using cooling channels built into the mould. The procedure is completed when the mould half (or halves) is opened and the solidified parts are ejected using ejector pins. This cyclic process is widely considered as a fast and efficient technology that allows for the creation of complicated geometries with intricate features. Numerous physical parameters control the entire process, and it has been established that there is a link between the process parameters of materials, part shape, and the quality of the moulded components.

In one study, cavity balancing was highlighted as a critical factor for improving the quality of moulded components during filling analysis. If there's an imbalanced flow pattern, it would cause packing issues and part warpage. In addition, choosing the right gate position can help reduce filling time and even out the temperature distribution of the moulded parts. Variations in shrinkage can also be caused by moulding process parameters. To prevent the creation of a frozen layer and warpage, the injection time, melt temperature, packing pressure, and packing time must all be chosen carefully. At higher temperatures, the melt shrinks more, but at lower temperatures, the melt shrinks less.

Component details

The component used for this study is ION base plate used in the assembly of an alpha ventilator system. The material used for the manufacture if this component is PC-ABS blend and the make is Bayblend 3010. PC-ABS is a blend of polycarbonate and acrylo nitrile

E-mail address: prekshithdude@gmail.co

International Journal of Research Publication and Reviews Vol (2) Issue (8) (2021) Page 721-727

butadiene. The material used for the manufacture is a amorphous thermoplastic. The selected material has increased heat resistance, improved chemical resistance and stress cracking behaviour and a good flame retardant. The material used for the manufacture is an amorphous thermoplastic.

Component name	ION Base Plate
Material	PC-ABS
Component Volume	179.46 cm^3
Density	1.18 gm/cm^3
Shrinkage	0.5-.7%
Melting temperature	270°C

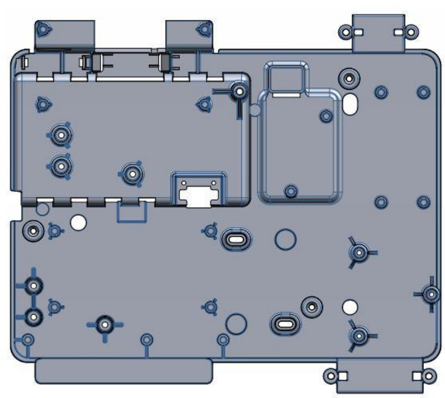

Fig. 1 Top view of the component

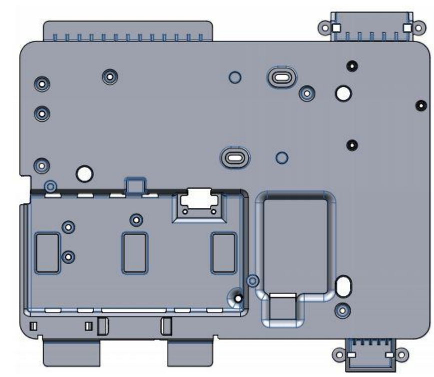

Fig. 2 Bottom view of the component

Analysis through Moldflow adviser

Moldflow Adviser from Autodesk is a small and affordable solution for simulating design during component creation. Development time and costs can be dramatically reduced with the digital prototypes that result. It allows the designer to use a CAD programme to create a model of the part, which is then read into the adviser. Some information must be input, such as the polymer and injection location. Temperature and pressure, for example, are selected automatically according on the material properties. It can, however, be tweaked to fit special requirements.

The following generated findings can be obtained after the analysis is completed.

- Best gate location
- Evaluation of fill time
- Confidence of fill
- Injection pressure required
- Locating the weld lines and
- Air traps
- Optimizing flow rates for cooling mould

Inputs given to the software,

Polymer type	PC-ABS
Producer	Bayer
Melt Temperature	270.000 ˚C
Mold Temperature	80.000 ˚C

2.1 Best gate location

The conclusion of the examination of the Gate Location is the Best Gate Location. This result confirms that every point on the model is injection-ready. The most suitable regions, which are shown in blue, are known to be the best locations for the component. The portions of the model that are least suitable for gate placement are marked in red. The best location rating does not always imply that the section can be filled in that place. Other factors have an impact on the filling, therefore precise results can only be obtained by running the study. The best gate location for the component is obtained by the analysis carried out.

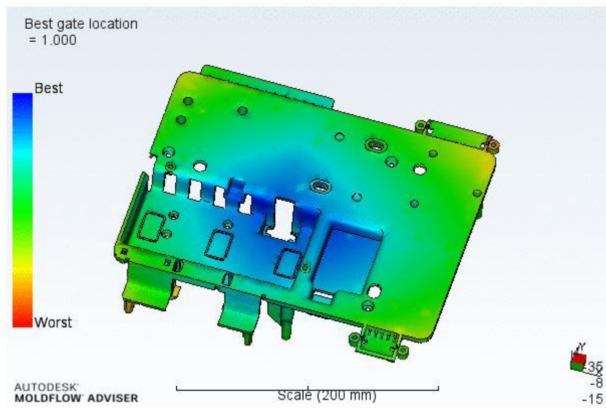

Fig.3 Best gate location

From this analysis, we understand that the sprue bush that is gate provided at the blue zone results in the better filling.

2.2 Fill time

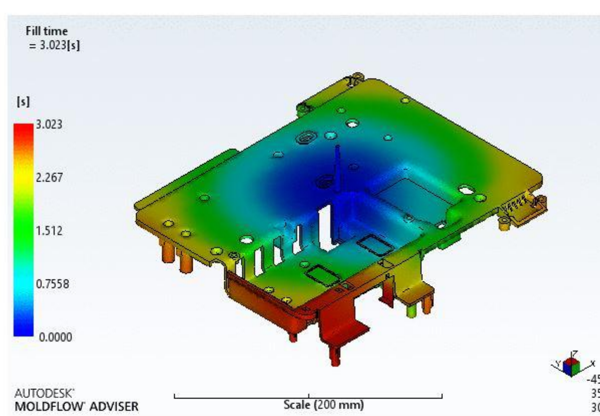

Fig. 4: Fill time

The plastic flow path through the part is depicted in this image, with each hue reflecting the portions of the mould that were filled at the same moment. A little shot will be presented as transparent. The fill time result is usually used to see if all of the flow paths fill at the same time and how much fill time is needed to fill the cavity completely. The fill time for the component is approximately 3.023 seconds, as seen in figure.

2.3 Confidence of fill

The confidence of the fill result represents the likelihood of a region within the cavity filling with plastic. The findings show model's green, yellow, red, and translucent parts. Where, the green color shows that the part can be filled completely and translucent represents the low fill area. With the chosen gate location the part shows the better filling ratio.

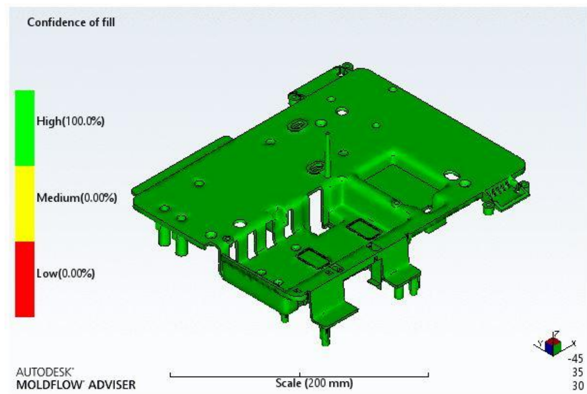

Fig. 5: Confidence of fill

2.4 Injection pressure

The reciprocating screw applies injection pressure to molten plastic resin to force it into a mould cavity that is up to 95% of its original size. It is governed by the size and shape of the parts, as well as the size of the door openings, and is evaluated against the machine's clamping pressure. When the part is completely filled, the colour at each position on the model represents the pressure at that location. At the end of the fill, this is a snapshot of the pressure throughout the entire part. From the analysis, the obtained injection pressure is 83.20 MPa

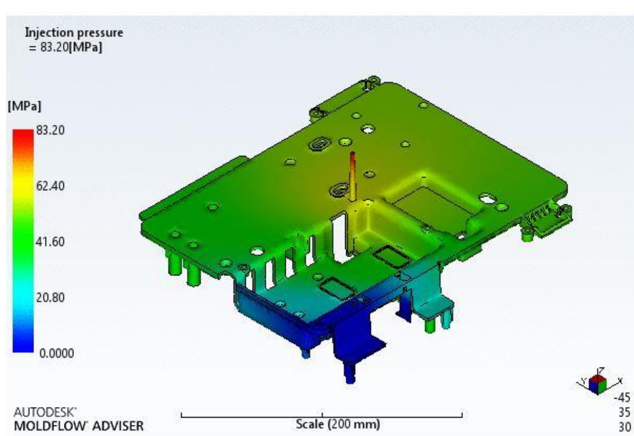

Fig. 6: Injection pressure

2.5 Weld lines

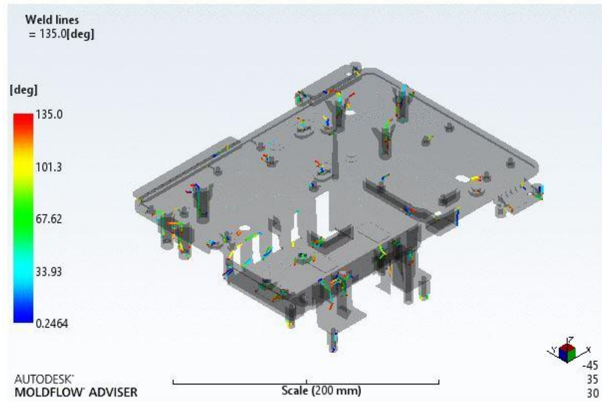

Fig. 7 Weld lines

A weld line is a defect on the surface of a moulded plastic item that appears as a mark. It usually takes the form of a line and can be found pointing away from a hole, inclusion, or other feature. Weld lines appear when molten plastic resin is pushed through the gate into a mould cavity. The cavity is filled at high pressure through the gate to accomplish total filling. During the cooling stages, the part comes into touch with the cavity walls, which are cooler due to the cooling channels. When resin is forced to flow around an obstruction, such as a hole or depression, the flow path splits into two channels, which reunite once the barrier has been crossed. During uneven cooling, the two flow fronts are forced together. The two sides are unable to fully merge because the plastic resin has already had time to come into touch with the cooler tool walls and as a result of this solidification, resulting in a distinct welding line. The results reveal that the anticipated weld lines are acceptable because they are located in the least sensitive location.

Weld lines should be avoided in areas that require strength or must appear smooth. To get a different fill time, change the polymer injection position or change the wall thicknesses. Flow fronts may meet at a different place with a different fill time, causing the weld/meld line to migrate.

2.6 Air traps

An air trap is formed when convergent flow fronts surround and capture a bubble of air. When flow paths are imbalanced or there isn't enough venting, this happens. As a result of the analysis, the areas of the air trap have been shown. The majority of air trap placements are appropriate because they are non-functional and out of sight.

Excess gas in the component, which might result from high melt temperature, low back pressure, or high decompression, is the most common cause of Air trap. Changing these parameters, and if possible, using short shots, could resolve the problem.

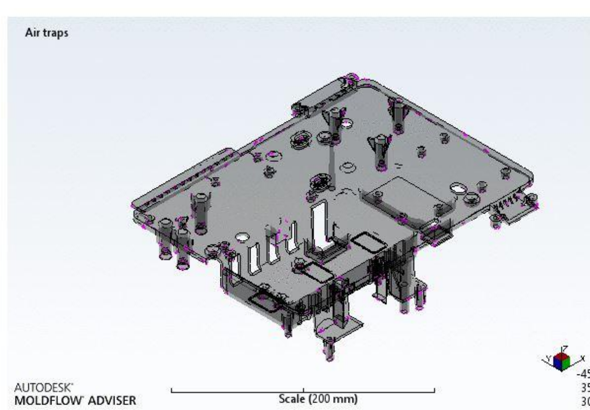

Fig. 7 Air traps

2.6 Flow front temperature

The melting temperature is another significant component to consider when moulding plastic items. The resin will not be properly plasticized if it is not plasticized or is too viscous to flow. If the melt temperature is too high, the resin may lose its properties and degrade. For particular materials, the resin manufacturer recommends melt and mould temperatures.

It shows how the temperature is distributed throughout the moulding. The findings also hint to a rise in temperature; if the temperature exceeds 310C, the material would burn and the part quality will degrade. The analysis in figure indicates that the temperature increase is not above the predetermined limit. As a result, the outcomes are satisfactory.

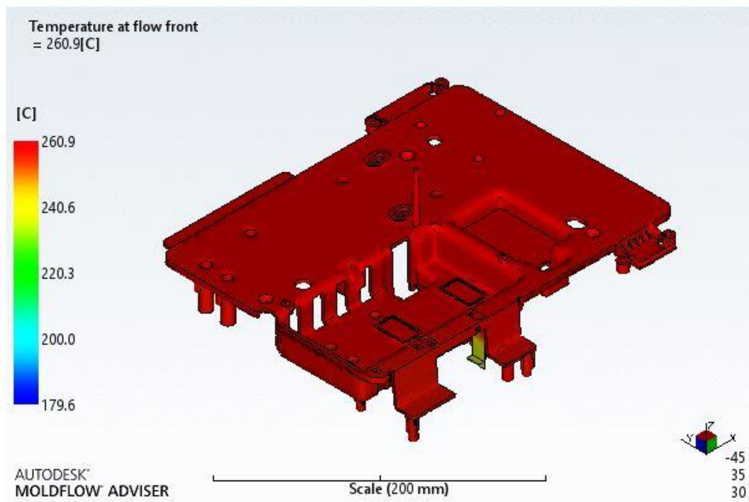

Fig 8 Flow front temperature

Discussion and Conclusion

From the analysis carried out for the component ION base plate using Autodesk mold flow adviser some of the important factors influencing the design of the mould is obtained. The obtained result is further used during the design of an injection mould tool for the optimized results.

- Suitable fill time is obtained as 3.2sec which is optimum based on the design and machine parameter.
- Weld lines locations are obtained and can be avoided by varying the injection mould parameters such as melt and mould temperatures.
- Air traps are eliminated by providing suitable air vents in the mould.
- Required injection pressure is obtained and can be used during the process.

REFERENCES

1) B. S. Lombardo, H. Keskkula, and D. R. Paul, "Influence of ABS Type on Morphology and Mechanical Properties of PC/ABS Blends " Department of Chemical Engineering and Center for Polymer, 1994.
2) Muralidhar Lakkannan, G.C. Mohan Kumar, Ravikiran Kadoli, "Computational design of mould sprue for injection moulding thermoplastics" Journal of Computational Design and Engineering 3 (2016) 37–52, 2015.
3) Sreedharan J and A.K Jeevanantham, "Optimization of Injection Molding Process to Minimize Weld-line and Sink-mark Defects Using Taguchi based Grey Relational
Analysis", Materials Today: Proceedings 5 (2018) 12615–12622, 2017.
4) O. Rashid, K.W.Q. Low, J.F.T. Pittman, "Mold cooling in thermoplastics injection molding: Effectiveness and energy efficiency" Journal

of Cleaner Production 264 (2020) 121375, 2020.
5) Marton Huszar, Fawzi Belblidia, Helen M. Davies, Cris Arnold, David Bould, Johann Sienz, "Sustainable injection moulding: The impact of materials selection and
Gate location on part warpage and injection pressure", Sustainable Materials and Technologies 5 (2015) 1–8, 2015.
6) Ke Yao , Furong Gao , Frank Allgo¨wer, "Barrel temperature control during operation transition in injection molding", Control Engineering Practice 16 (2008) 1259– 1264, 2008.
7) Jian-Yu Chen, Kai-Jie Yang, Ming-Shyan Huang," Online quality monitoring of molten resin in injection molding", International Journal of Heat and Mass Transfer 122 (2018) 681–693, 2018.
8) Isaac Meekers, Paul Refalo, Arif Rochman, "Analysis of Process Parameters affecting Energy Consumption in Plastic Injection Moulding" Procedia CIRP 69 (2018) 342 – 347, 2018.
9)Jan L. Spookerman, "The role of failure analysis in establishing design rules for reliable plastic products", Microelectron. Reliab, Vol. 35, Nos 9-10, pp. 1275-1284, 1995.
10) D.E. Dimla, M. Camilotto, F. Miani, "Design and optimisation of conformal cooling channels in injection moulding tools", Journal of Materials Processing Technology 164–165 (2005) 1294–1300, 2005.