ISSN NO. 2320-5407

Journal homepage: http://www.journalijar.com

INTERNATIONAL JOURNAL OF ADVANCED RESEARCH

RESEARCH ARTICLE

FLOW ANALYSIS OF BOTTOM CASE

*** Rahul S. Khichadi**
Vishwabharathi academy's college of engineering, Ahmednagar.
University of Pune, Maharashtra, India- 414201

Manuscript Info

Manuscript History:

Received: 15 December 2013
Final Accepted: 20 January 2014
Published Online: February 2014

Key words:
Analysis; Gate Location; Fill Time

***Corresponding Author**

Rahul S. Khichadi

Abstract

Injection molding is a net shape manufacturing process in which a polymer melt is forced into an evacuated mold cavity that cools the polymer melt into a desired shape. Injection molding has been a challenging process for many manufacturers and researchers to produce products meeting requirements at the lowest cost. Moldflow Plastics Insight software represents the most comprehensive suite of definitive tools for simulating, analyzing, optimizing, and validating plastics part and mould designs. It gives detailed information about the processing conditions. It has now become a prerequisite to validate the design and reduce the iteration involved in trial and error method of manufacturing tools. The lead time is also drastically reduced. This paper interprets the result done on gate analysis, fill analysis using MPI software. In this paper, injection molding process analysis is presented for Bottom case of Energy meter assembly.

1.Introduction:

The injection molding process is primarily a sequential operation that results in the transformation of plastic pellets into a molded part. Identical parts are produced through a cyclic process involving the melting of a pellet or powder resin followed by the injection of the polymer melt into the hollow mould cavity under high pressure.

Moldflow Plastics Insight (MPI) software which is an integrated suite of analysis tools that utilize CAD files and apply advanced Finite Element Analysis (FEA) techniques to quickly and easily enable a virtual "what if" design environment before initiating mould construction. MPI provides in-depth part/mould design and process parameter optimization.

The mold flow plastic Insight is designed to help and determine the filling capacity of plastic part design. The mold flow allows one to create a model of part in a CAD package, and then read the model into the mold flow. The MPI supports stl Format files taking from higher end CAD software.

One needs to input information, including the polymer and the injection location. Other settings, such as material temperature and pressure, are automatically selected according to the materials properties. But it can be altered according to the desired one.

2. Experimentation

Here we carry out analysis on Bottom case of an Energy meter as an example Purpose of this analysis is for optimizing the injection molding process. The component name is Bottom Case. The component goes into a device which is used for Energy meter. The weight of component is 57.04cm^3.The plastic material used to make this component is ABS

2.1 Study of the Component

2.2 Material (ABS)

ABS is a amorphous thermoplastic, which is tough and dimensionally stable and it has 0.005-0.007% of shrinkage and with solid density of 1.0541gm/cm^3.It is used in applications that require high-performance properties. It is a material having good chemical, electrical & weathering properties. For this component electrical & weathering properties are very much required.

3. Flow Analysis

Moldflow Plastics Insight (MPI) software is used to analyze plastic flow. The MPI represents the most comprehensive suite of definitive tools for simulating, analyzing, optimizing, and validating plastics part and mould designs. **Moldflow is an outstanding predictive engineering tool. It's a simulation tool which can identify potential problems or address uncertainty. The scientific approach used will save both time and money.** To eliminate uncertainty and to insure the design can be moulded successfully. Moldflow analysis is a "must have" for reducing costs and optimizing productivity.

3.1 Gate location

Xiaoyan and Qiang [6] discuss importance of gate location; the gate location represents the position where polymer is injected. In order to mold the best part possible, here we have to decide the optimum injection location for our part. The optimum injection location creates uniform flow. The injection location represents the best possible position where the polymer has to be injected in order to get the components of an acceptable quality level.

Here we are doing analysis for single cavity. The position of best Gate is generated by the Best Gate Location analysis. The Gate Location result rates each place on the model for its suitability for an injection location. Mold flow has color code for pointing out the suitable location for the gate. The most suitable areas are rated as best (blue colored), and the least suitable areas of the model are rated as worst (red colored). Here in fig 3.1 show best gate location and shows maximum acceptable gating position. Because it's quite difficult to form gate in blue color according to manufacture point of view, so gate location made max acceptable gate position that is in sky blue color. Based on the analysis the designer would be able to fix up the gate location

3.2 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 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.

- All green – The part is easily molded and part quality is acceptable
- Blue – This region shows easily filled part.
- Yellow – The part may be difficult to mould or quality may not be acceptable
- As the percentage of yellow increases, the difficulty in molding the part will increase and the part quality will decrease.
- Red – The part will be extremely difficult to mould, as the quality may give resistance to fill.
- Translucent – the part cannot be molded because short shot will occur.

3.3 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. The total fill time indicated as 1.136 seconds is on the red zone. Which indicates this can be reduced by varying the runner parameter.

3.4 Freeze time

The Time to freeze result shows the time taken for the part to freeze to ejection temperature, measured from the start of the cycle. This result shows in fig 3.4, the freeze time of the plastic throughout the entire sprue and component parts. Its complete process of freezing is within 56.30 (Red color) seconds; this freeze time has to be reduced by changing cooling parameters.

3.5 Temperature at flow front

Dubay, Pramujati and Harnandez [7] said the controlling of cavity temperature for best analysis. The temperature at flow front result uses a range of colors to indicate the region of lowest temperature (colored blue) through to the region of highest temperature (colored red). The colors represent the material temperature at each point at which it has been filled. The result shows the changes in the temperature of the flow front during filling. The result shows in fig 3.5. The temperature of part can be altered by giving proper cooling arrangement. The temperature at flow front in the most region of the component varies from 227.4 to 230.4^{0}C which is close to the melt temperature of the polymer (221^{0}C).

3.6 Air Traps

The air traps result is generated in flow analysis, and Fig 3.7 shows pink dots wherever an air trap is likely to occur. This is eliminated by providing suitable air vents in the mold.

The Figure 3.7 shows possibility of air traps occurrence in main phase of component in order to overcome this problem below mentioned are solution adopted during tool design process.

- By providing Air vents of order 0.03mm in Core & cavity matching surface, airtraps are minimized.
- Clearance in Ejector pin of order of 0.02mm per side is provided.

3.7 Weld Lines

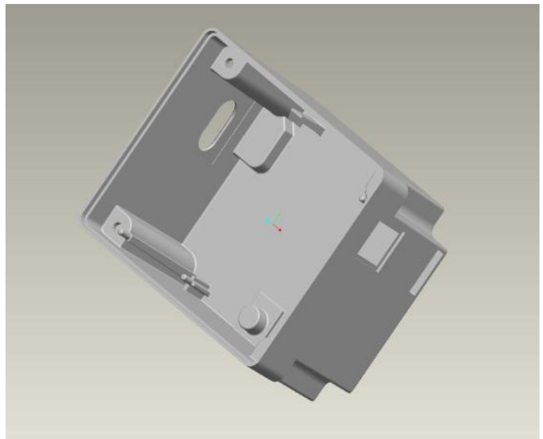
When a weld line forms, the thin frozen layers at the front of each flow path meet, melt, and then freeze again with the rest of the plastic. The orientation of the plastic at the weld is therefore perpendicular to the flow path.

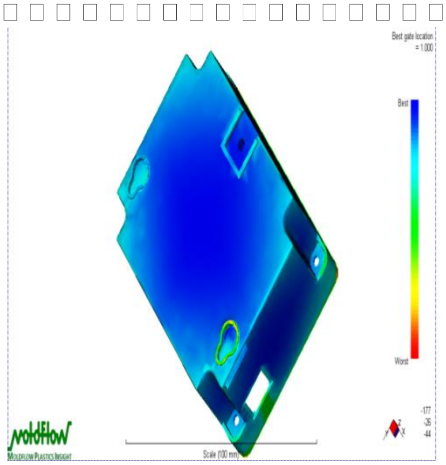
This result indicates the presence and location of weld and meld lines in the filled part model. These are places where two flow fronts have converged. The presence of weld and meld lines may indicate a weakness or blemish. Shayfull [8] discusses reduction of weld lines and he mentioned it’s difficult to avoid weld lines completely. So formation of weld lines is acceptable.

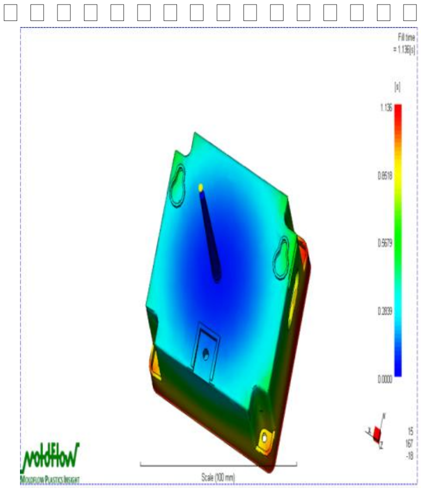
4. Results and discussions

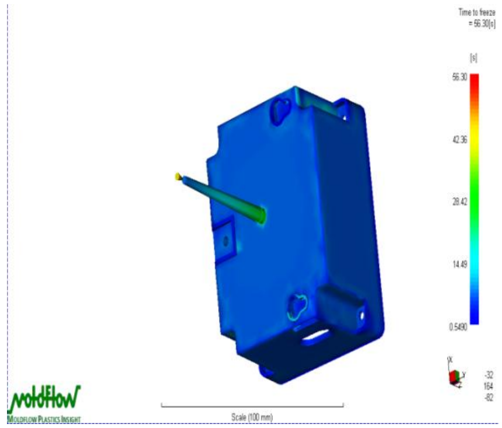
After conducting the above mentioned analysis we have got the following results.

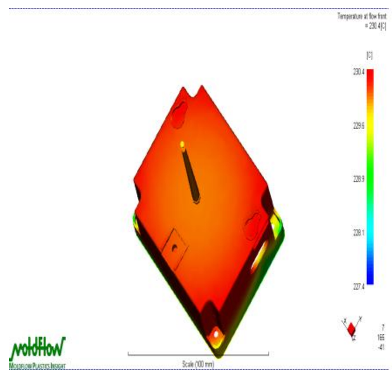
- Best location of the gate obtained from the analysis has been selected for manufacturing and in design. The present gate size and location is finalized after much iteration from the point of uniform filling and other factors.
- The fill time is 1.136 sec, which is reduced by varying runner parameters.
- The temperature at flow front in the most region of the component varies from 227 to 230.4^{0}C which is close to the melt temperature of the polymer (221^{0}C), so formed weld line has little effect on the strength of the product.
- The temperature near the weld line is around the 230˚C, which is very close to melt temperature, which results weld lines to fuse.
- Most of the air traps are located on the surface, which can be removed by providing suitable air vents in the mould.

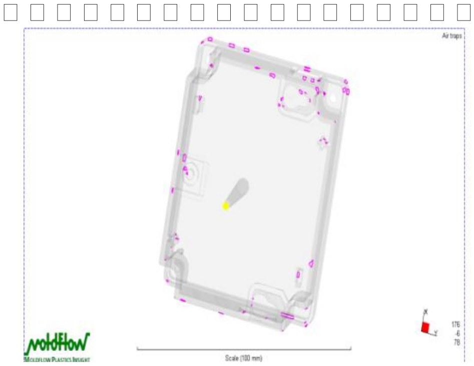


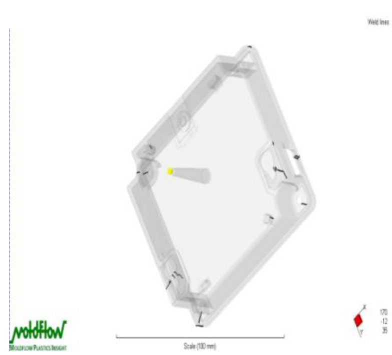

Best gate location
= 1.000
Best
Worst
Scale (100 mm)


Fill time
= 1.136[s]
[s]
1.136
0.8518
0.5679
0.2839
0.0000
Scale (100 mm)


Time to freeze
= 56.30[s]
[s]
56.30
42.36
28.42
14.49
0.5490
Scale (100 mm)


Scale (100 mm)


Air traps
Scale (100 mm)


Weld lines
Scale (100 mm)

5.Conclusion

In the present study, the simulation for the Bottom case has been performed by Moldflow software. The best gate location was obtained in this work, which ensures the product can be filled completely. Through the comparative analysis of gate, the sprue gate is acceptable according to the distribution of air trap, the weld line and volume shrinkage.

References

[1]JagannathYammada,Terrence L. Chambers, Suren N. Dwivedi. Intelligent Tool for Plastic Injection Mold design. Volume 7 Issue 2(2004).

[2] Iwan Halim Sahputra. Comparison of Two Flow Analysis Software for Injection Moulding Tool Design(2007).

[3] Zhi-Xin Chen, Zhi-Hui Wan.
Comprehensive Simulation Analysis of Plastic Injection Process(2009).

[4] Zhao Longzhi, Chen Binghui , Yang Min, Zhang Shangbing. Application of Moldflow Software in Design of Injection Mold(2010).

[5] Jian Zhu, Chuanyang Wang. Flow analysis in Injection Molding Process Based on Moldflow.

[6] Huang Xiaoyan, Xu Qiang. Optimization Model of Injection Gate Location Based on Empirical Method, International Forum on Information Technology and Applications, p 428-430(2009).

[7] Rickey Dubay, Bambang Pramujati and Jose Hernandez. Cavity Temperature Control in Plastic Injection Molding, International Conference on Mechatronics & Automation.
Niagara Falls, Canada, p 911-916 (July 2005).

[8] Z. Shayfull. Optimizaton of weld line formation by taguchi approch and analysis of variance(2011).