Injection Mold Design and Optimization of Battery Air vent

Rahul S. Khichadi
M.Tech student, VACOE Ahmednagar, Maharashtra, India-414201

Abstract—In this paper, battery vent plug part for injection molding process analysis and die design is presented. Injection molding part which is used in battery for the venting purpose is modeled and analyzed to determine the parting surface, gating systems. Injection molding parts conduct mold flow analysis to determine the feasibility, and the die gating system and the injection molding process are optimized. Thus, high projection quality products are obtained. To permeate this , the use of Computer aided design (CAD) and Computer aided engineering (CAE) application assisted in design and establish the most optimal parting surface and gating system. This research was conducted by implementing the use of CAD/CAE tools to produce an optimal mold design using Pro/E and Moldflow® applications. The application tremendously assist the designer in resolve the mold with part surface and best gate location in producing an battery part with low defects, fast time-cycle and the most important factor is to reduce the manufacturing cost.

Key words – computer aided design(CAD); Computer aided engineering(CAE); Analysis

I. INTRODUCTION

With more and more extensive application of plastic products, plastic mold industry has been rapid development. However, plastic products have diversity, complexity and limitations of experience, it is difficult to accurately design a set of lower costs, save time in a long time, improve product quality and the passing rate of the most reasonable mold process scheme [1]. In the design of mold, there are some things which to be attention when doing design of mold[2], that are shape of product, cavity, runner system, gates system[3],cooling channels[4]， venting of mold and heat exchange system[5]. Because of that, if a design have made, need to depend on the things in above which has been part of mold design standard. However, it is decided for the type and condition in above that the most of fit with the shape of product is very difficult. Example, for decide the location of gate, if location of gate that has been decided not be good, so the result of product could be not maximum. To overcome this problem, simulation is made before the mold is manufactured, to improve the quality of result after injection depend on design that would to be made, so if the mold was made could be result a good product. Software would be used is Mudflow®. When use this software, we can predicted many things concerning with injection process and can improve quality of product that resulted in the injection process. Mold work often process flow analysis model to get a good structure and injection mold parameters, and become really involved in pre-production mold structure as much possible to solve the problems, support mold, designers to design accurate and efficient mold, mold design to improve quality, reduce mold costs, shorten delivery schedules mold, which has been widely used. In this paper, battery air vent mold as an example is illustrated mold design applications by Moldflow analysis.

II. TECHNOLOGY ANALYSIS AND IDENTIFICATION

Battery air vent Plug is major component of product in battery interiors. The surface quality requirements are high, and product surface appearance of surface requires a good process of forming. At the same time in order to increase product strength, lower die cost, improve product quality. Therefore, the focus of analysis is divided into filling flow analysis and injection molding analysis.

The Battery air vent component is shown in Fig.1, and its material is Polypropylene(PP), adopt the mode of production in enormous quantities. It is about Ø38.5x22.5 mm to mold a whole size, and there are undercuts and thin section on inner surfaces of the part, make the structure of the mold relatively complicated, need to carry on comparatively careful treatment to these little structure in the design, has increased the degree of difficulty of mold design. In order to meet the requirements of mold design, the way can be selected to realizes the split cavity mechanism with sleeve ejection.

Fig.1 Parts diagram of the Battery air vent plug

III. EXPERIMENTATION

Here we carry out analysis on Battery air vent plug as an example. Purpose of this analysis is for optimizing the injection molding process.

Injection molding temperature	200º C
Mold temperature	80º C
Runner (Trapezoidal in each cavity)	30mm length, angle 16º and diameter Ø4mm
Gate	Tapered 1.6x1.1x0.5 mm

3.1 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.

3.1.1 Gate Location

Gate location have great influence on the results of the moldflow analysis. This is because the Battery air vent structure, there are many ribs and melt flow is poor. On the basis of considering gate location as even as possible symmetrical distribution, can be successfully filling to ensure that products of the physical and mechanical properties. Battery air vent plug outer surface is not required to be the high quality one, as it is not exposed to outer environment. Gate can be made at the parting surface were the two cavity surface meets[5]. To simplify the modeling, to improve analysis efficiency, optimize the design of gate location, the Battery air vent plug will be analyzed with Moldflow®. The best gate location analysis of the gate is shown in Fig.2. The outer surface of the blue part in the figure can be seen as the best gate location, and thus the gate should first consider setting up in the middle of the blue part. Since there are two best gate locations on top and the middle of the component. The gate location on the larger diameter is considered. Since for single cavity the manufacturing cost is more as compared to two cavity mold and the balancing of the mold is very difficult, so to avoid these problems we have gone for two cavity mold.

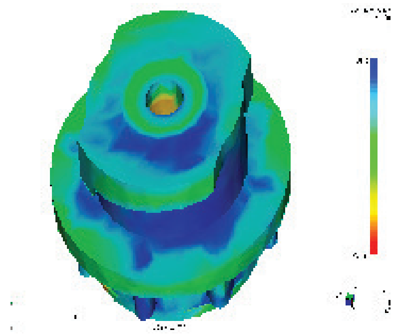

Fig.2(a) . The best gate location analysis for single cavity

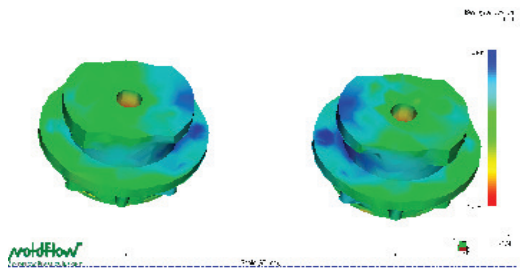

Fig.2(b). The best gate location analysis for two cavity

The best gate location analysis is a more important link in Moldflow analysis, which is an important reference to design the gate location of a multi-cavity mold. Results with different colors display the best gate location which is blue area, as shown in Fig.2(b). According to the analysis of the Tab.1 Moldflow analysis parameter settings best gate location in Fig.2(b), The gate location on the larger diameter is considered and then filling and cooling analysis are proceed. During the process of moldflow analysis, the gate size, temperature and other parameters to set the analysis parameters are preliminary estimated. The analysis is primarily aimed at filling analysis to predict whether the gate is set and size to meet the requirements and possible defects location in parts are analyzed. Parameter setting is in Tab.1.

3.1.2 Filling Analysis

Filling analysis should be compared results of the analysis through the different flow behavior of the system to choose the best gate location, the best layout. Meanwhile, the workpiece should be avoided to infilling injection, as well as imbalances in flows and so on, to avoid or reduce cavitations and weld lines, and as far as possible a lower injection pressure, clamping force, to reduce the performance parameters of injection molding machine requirements, to make flow analysis as much as possible to avoid or reduce shrinkage caused by improper packing of the product, warping, distortion and other quality defects. As can be seen from Fig.3, the filling time is 2.136s. The first part of the component is filled with blue position, the last part of it is filled with the red figure position in the cooling water should be set up as far as possible away from the red area.

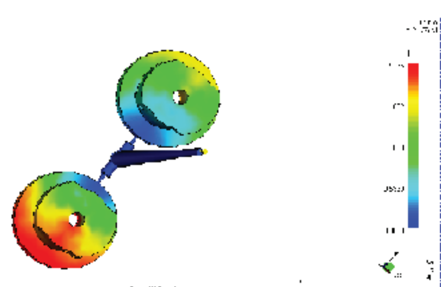

Fig.3 Filling analysis

3.1.3 Pressure

When the filling mold finishes, the pressure figure for observing is showed that the distribution of component is balanced. Because at the end of filling pressure on the balance is very sensitive, so if the pressure distribution figure is balance at this time, then the component achieve a good balance on the filling. According to Fig.4, the finishing filling pressure distribution show that in the last blue part filled is lower part of the pressure, and the entire injection molding parts pressure difference is below in 19.89MPa

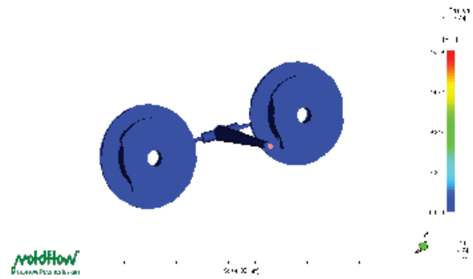

Fig.4 pressure

3.1.4 Temperature at flow front

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 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 211.5 to 233^{0}C which is close to the melt temperature of the polymer (230^{0}C), so formed weld line has little effect on the strength of the product.

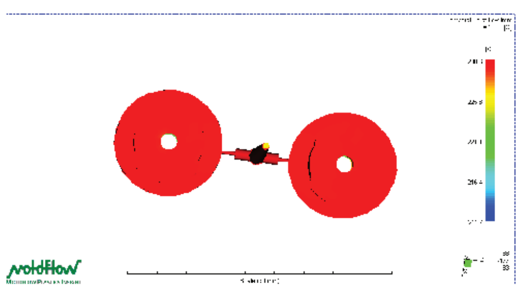

Fig.5 Temperature at flow front

3.1.5 Weld Lines

When two or more of the flow front integration, it will form a weld line. In the weld-line position the molecules are tend to change strongly, so that the mechanical strength significantly weakened in the position, and weld line is not visually obvious. In this case, increasing the mold temperature and melt temperature make two melt encounters merge better. It is advisable to increase the screw speed or improve the design of gating system to reduce the friction generated heat, while maintaining melt flow rate to reduce flow channel dimensions. Flow front temperature is the middle temperature when the polymer melt packs a node. It represents the temperature of the sectional centre, so it does not change much. Fig.4 shows that when the material fusion temperature is around 230 ℃, the lowest melt temperature is only lower than the injection temperature of 211.5 ℃. So materials can be a very good fusion, and does not appear Weld. Cavitations are seen in the fusion line and parting surface, and exhaust ducts are set at the opening of weld line. This is not only convenient to exhaust, but also increase the fusion line of fastness. The weld line is not obvious, and the results are best.

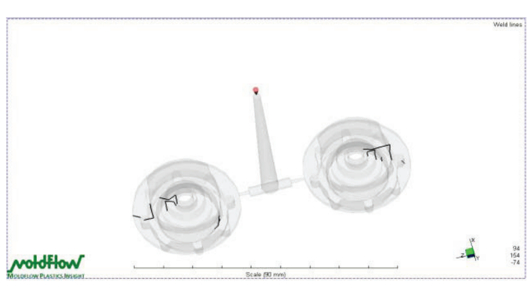

Fig6. Weld-line

3.1.6 Air traps

The air traps result is generated in flow analysis, and Fig 7 shows pink dots wherever an air trap is likely to occur. This is eliminated by providing suitable air vents in the mold.

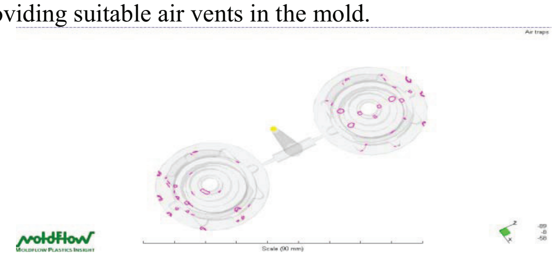

Fig.7 Air traps

Above Air Traps Results

- Burn marks
- Short shot
- Other surface blemishes

The Figure 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.
- Air Traps has been found at the component corner`s i.e. at top and bottom of the component after injection molding.

3.1.7 Freeze time

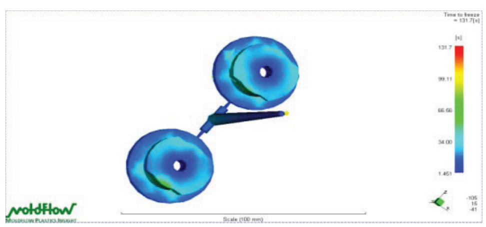

Fig.8 Freeze time

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 8, the freeze time of the plastic throughout the entire sprue, runner, gate and component parts. Its complete process of freezing is within 32 (Red color) seconds; this freeze time has to be reduced by changing cooling parameters

IV. 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 2.136 sec, which is reduced by varying runner parameters.
- The temperature at flow front in the most region of the component varies from 211 to 233.4^{0}C which is close to the melt temperature of the polymer (230^{0}C), so formed weld line has little effect on the strength of the product.
- The temperature near the weld line is around the 233˚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.

V. CONCLUSION

- In the present study, the simulation for the Battery air vent Plug 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, runner, gate is acceptable according to the distribution of air trap, the weld line and volume shrinkage.
- There may be chance of reduction in total cycle time. This can be achieved by certain experiments.

REFERENCES

[1] A B M Saifullah and S. H. Masood, Optimum cooling channels design and Thermal analysis of an Injection moulded plastic part mould, Materials Science Forum, Vols. 561-565, pp. 1999- 2002, (2007). *(references)*

[2] Bingfeng, F.; Kazmer, D.; D. Hatch, 2001, "Simulation based optimization of injection molding", Polymer Processing Society Conf.

[3] J. Koszkul, J. Nabialek, Viscosity models in simulation of the filling stage of the injection molding process, Journal of Materials Processing Technology 157-158 (2004) 183-187.

[4] S. Jones, M.R. Jolly, S. Blackburn, J.-C. Gebelin, A. Cendrowicz and K. Lewis,"Measurement of Autoclave Thermal Profiles During High Pressure Steam Dewaxing of Investment Shells Part 2: Wax Body Profiles", Journal of Material Science and Technology

[5] D.E. Dimla, M. Camilotto, and F. Miani: Design and optimization of conformal cooling channels in injection moulding tools, J. of Mater.Processing Technology, 164-165,pp1294-1300