

Analysis of Plastic Component Using Pro/E

Mohit N. Rane¹, Tejas I. Chaudhari², Jitendra D. Koli³

¹Pune University, MMIT, Department of Mechanical Engineering, Lohgaon, Pune 411047, India

²North Maharashtra University, SSGBCOET, Department of Mechanical Engineering, Bhusawal 425201, India

³Pune University, DYPCOE, Department of Mechanical Engineering, Akurdi, Pune 411035, India

Abstract: With increasing demand in plastic products the plastic industries are growing in faster rate with requirements are intricate shapes with good dimensional accuracy, to meet such a requirement it is important to adopt various advance technologies like CAD/CAM/CAE for development of injection molded components. In this paper, an analysis has been performed by taking plastic indicator of Hero Honda CDI100. Mold Flow Plastic Advisor (MPA) simulation tool from Pro/E used for analysis to optimize the different parameters which considered while actual manufacturing of product. This process requires less time to analyze the different parameters such as confidence of fill, best gate location, molding defects etc. with no material waste as compared to conventional hit and trial method used in production floor.

Keywords: Pro/E, Mold Flow Plastic Advisor (MPA), molding defects, Indicator.

1. Introduction

Now a day, the technology of the Tool and Die manufacturing is one of the fastest growing industries in the world. Plastic is the most used material in the world. Among various plastic production technologies. In injection molding, the Tool Design of Mould is of Critical importance for the quality and economy. Injection molding is a process that forms the plastic into desired shape by melting the plastic material and forcing the plastic material under pressure into the mould cavity. Now a day there is a need for improving the process to increase productivity, reduce the cycle time in injection molding. In order to achieve the processing parameters, they commonly follows on experience, hit and trail method due to which this process is not practical for complex models. As new generation designers require more vital software to analyze and to optimize injection molding process by manipulating parameters to reduce cycle time. However development of CAD/CAM/CAE technology especially Mould flow Analysis, the number of trails on mould can be reduce to achieve good quality product.

2. Methodology

The method of doing analysis of component using Pro/E, is quite simple and has certain steps to followed,

- 1) The model/ component are created in software using commands.
- 2) Then the model is open in the part advisor for further analysis.
- 3) The plastic material is selected from directory available in software along with supplier.
- 4) Then gate locator is run by software which can be automatically selected or manually selected. The gate location is defined for mould flow analysis. After that analysis is to be run and then result is generated.
- 5) Look at the result, if the model will fill easily and create good quality of parts.
- 6) If the result is not satisfactory to fill the model then change one parameter and then reanalyze. These

parameters are,

- Position of injection location
- Mold temperature
- Injection time
- Parts Geometry.

- 7) Create a report using the Report Generator.

3. Analysis

Plastic advisor is an addition on analysis package for Pro/Engineer, especially for plastic injection molding. For doing this analysis, after drawing the required object Select applications > plastic advisor > pick datum point for injection location > ok.

There will be an analysis tool bar,



Figure 1: Analysis Menu in Pro/E.

The icons available are,

- Molding Parameter icon
- Specify Injection location
- Special Analysis Icon
- Start Analysis Icon

When we click on molding parameter icon a separate window opens and there we can select the required material and we can specify the required injection conditions like mould temperature, injection pressure etc.

After specifying the material and processing conditions click on the specify location for injection icon and specify the required location point then click on run analysis and select plastic flow analysis and click on start button
Results obtained are,

- Plastic Flow
- Fill time
- Confidence of fill
- Injection Pressure
- Pressure Drop
- Flow front temperature
- Quality Predictions
- Weld lines and air traps.

4. Analysis Report

The result for given plastic component is as follows,



Figure 2: Plastic Indicator part which is analyzed.

Table 1: Generated Analysis Report

Part Name:	indicator
Part Revision:	1
Material Supplier:	LG Chemical
Material Grade:	ABS MP220N
Max Injection Pressure:	250.00 MPa
Mold Temperature:	60°C
Melt Temperature:	235°C
Model Suitability:	Part model was highly suitable for analysis.
Confidence:	High
Injection Time:	0.55 sec
Injection Pressure:	29.34 MPa
Weld Lines:	Yes
Air Traps:	Yes
Shot Volume :	19.51 cm ³
Filling Clamp Force:	7.24 tonne
Packing Clamp Force Estimate @20%:	(5.87)MPa 2.44 tonne
Packing Clamp Force Estimate @80%:	(23.47)MPa 9.75 tonne
Packing Clamp Force Estimate @120%:	(35.20)MPa 14.62 tonne
Clamp Force Area:	40.73 cm ²
Cycle Time:	13.67 sec
Warnings:	No Warning
Sinkability:	3% of your model was found to be prone to Sink marks.

The results obtained in graphical format are as follows,

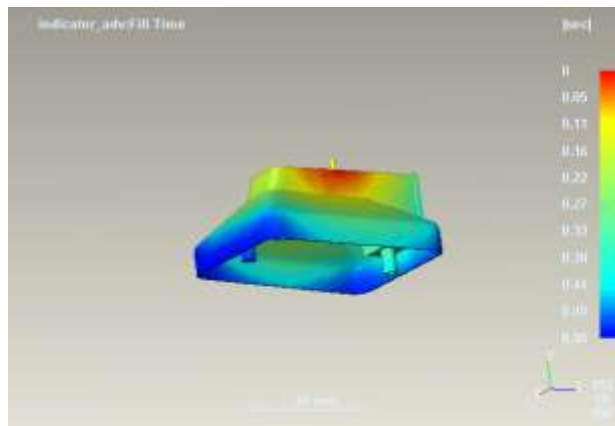


Figure 3: Fill Time

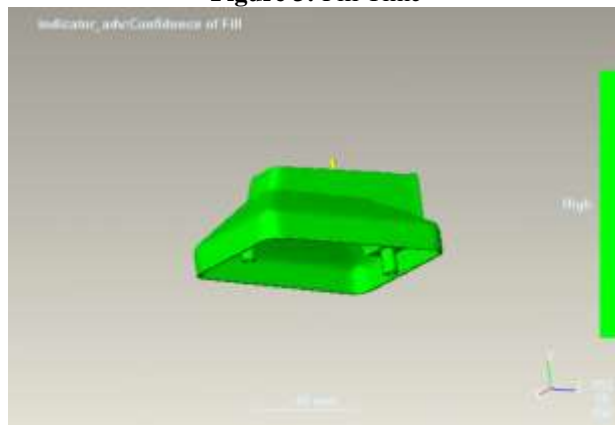


Figure 4: Confidence of Fill

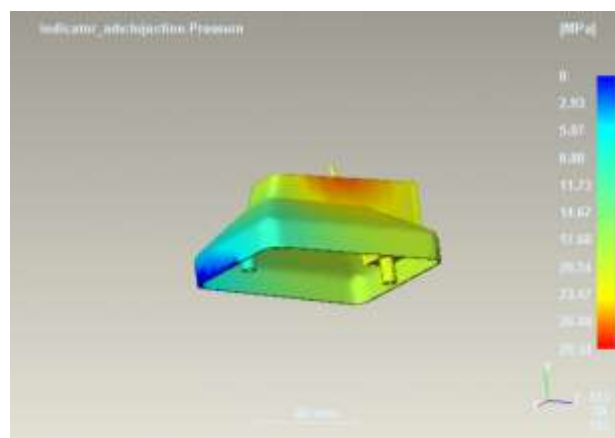


Figure 5: Injection Pressure

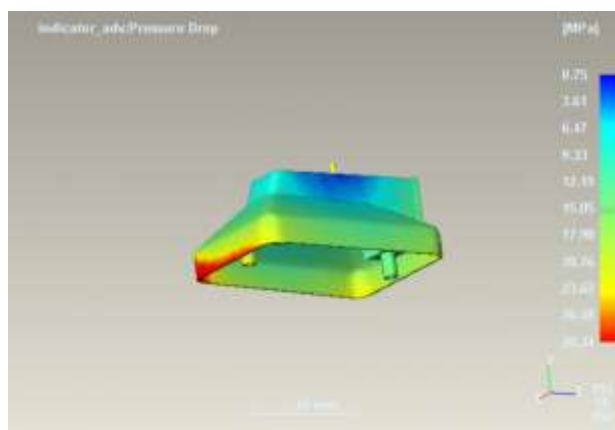


Figure 6: Pressure Drop

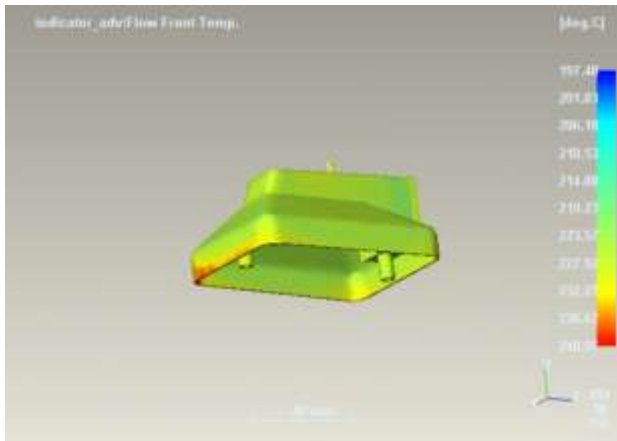


Figure 7: Flow Front Temp

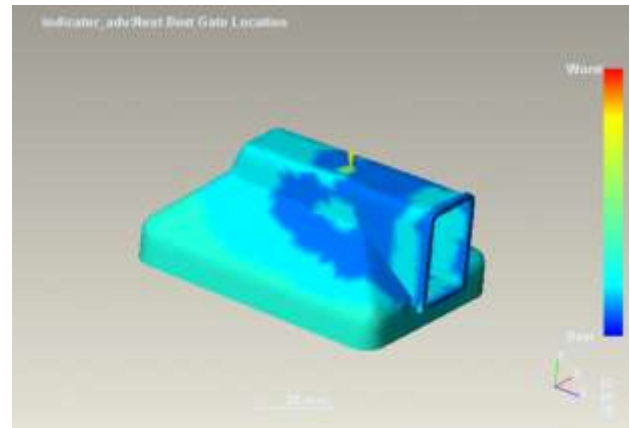


Figure 11: Next Best Gate Location

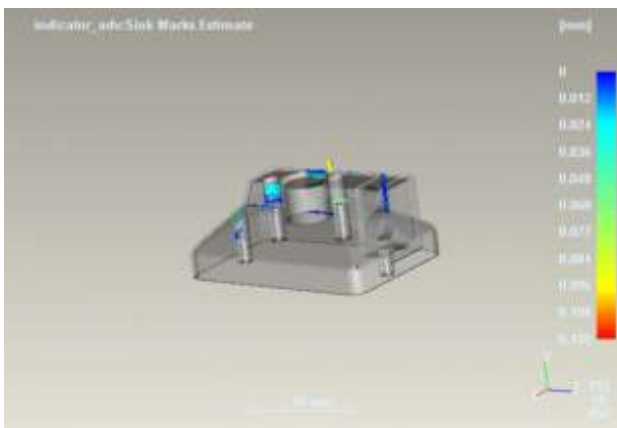


Figure 8: Sink Marks Estimation

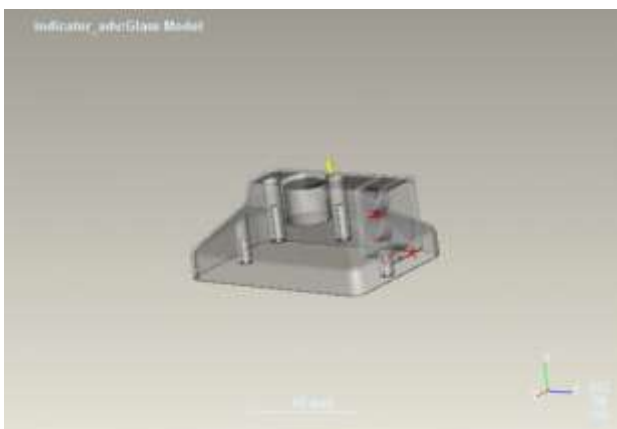


Figure 9: Weld lines



Figure 10: Air Traps

5. Result and Discussion

The Result generated by part advisor shows the various parameters of component and process details required to fill the mold. The advisor found the parameter like weld lines and Air traps which can affect the surface qualities of component. The weld lines indicates the presence of weld marks in filled part model and Air traps result shows region where melt drops forms due to bubble of air becomes trapped.

- Causes of Weld Lines
 - a) Low temperature of molding machine barrel
 - b) Inadequate back pressure
 - c) Injection Speed and Pressure is too low
- Remedies for Weld Lines
 - a) Change the wall thickness
 - b) Increasing melt temperature
 - c) Increasing injection rate and pressure
- Causes of Air Traps
 - a) Improper flow of molten plastic through mold
 - b) Due to sudden sharp corners or edges
- Remedies for Air Trap
 - a) Balance flow Path
 - b) Avoid hesitation and racetrack effect balance runner
 - c) Change the runner system

6. Future Scope

Today's world is changing day by day and modifications are done on every next second so, the future scope of this project title has large expansion. In this project we analyzed very small part and in this complex world short analysis and design for new product we always required improvement in this section. Also, as we know that the software analysis is done without use of actual part and analysis done with power of imagine and with low cost consumption so, future must relive with this software and we are assured that the next project will also give the same thing that future work should be done and it is just the beginning.

7. Conclusion

Using Pro/E and mold flow plastic adviser one can analysis the plastic component easily and using the obtain result as a guideline to develop the mold which is time saving as well as

the cost saving tool. One can go through the process that first create the model as per the requirement using Pro/E and then analysis the component using mold flow part adviser. From above all the result we predict the component is suitable for molding and as it shown in above mentioned picture there is very small defect which depend on processing condition. So the part can easily moulded by using ABS MP220N material From the above analysis the alternative gate location for better result, secondly if the design of mold is single cavity, multi cavity, or family any one of above condition the gate location plays major role for design of any mold cavity.

References

- [1] B. Pavani, M.V. Varalakshmi, "Mold Flow Analysis of Fan Part using Plastic Advisor", IJOER, vol-2, issue-7, july2016, ISSN 2395-6992
- [2] P. Hussain Babu, T. Vishnu Vardan. "Computer Simulation for Finding Optimum Gate Location in Plastic Injection Molding Process" International Journal of Engineering Research and Application, ISSN 2248-9622, Volume 3, pp 947-950.
- [3] Anumandla Madhukar, V. Narsimha Reddy, "Design and Optimization of Gate Location Mould Design and Cost Estimation of Base Plate EMF Load Cell" International Journal of Engineering Research-Online, Volume 3, ISSN 2321-7758
- [4] Pankaj Shakkarwal and Lipin Yadav, "Design and Mould Flow Analysis of Injection Mould For Connecting Link" International Journal on Emerging Technologies 4(1) 182-185 (2013) ISSN 0975-8364.
- [5] Wong C. T., Shamsuddin Sulaiman, Napsiah Ismail and A.M.S. Hamaida, "Design and Simulation of Plastic Injection Moulding Process" Pertanika J. Sci. And Technol. Supplement 12(2), ISSN 0128-7680, pp 85-99
- [6] B. Iftekhhar, Mir Safiulla, Mohamed Ali, G. Suresh. "Injection Mould Tool Design of Power Box Side Panel" International Journal of Innovative Research in Science Engineering and Technology, Volume 3, ISSN 2319-8753.
- [7] Yuefeng Li, "Design of Headphone Shell Injection Mold Base in Pro/E" Advance Material Research, vols 816-817, pp 307-310, ISSN 1662-8985

Author Profile



Mohit N. Rane pursuing the Bachelors in Mechanical Engineering from Pune University, India. Currently in final year of Engineering. Research interest includes Design analysis, Quality Improvement etc.



Tejas I. Chaudhari pursuing the Bachelors in Mechanical Engineering from North Maharashtra University, India. Currently in final year of Engineering.



Jitendra D. Koli pursuing the Bachelors in Mechanical Engineering from Pune University, India. Currently in final year of Engineering.