ANALYSIS AND DESIGN OF MOLD FOR PLASTIC SIDE RELEASE BUCKLE USING MOLDFLOW SOFTWARE

H Adithya Bhat[1], Sandeep Subramaniam[2], Arvind K. Pillai[3], Edu Krishnan L[4], M. Elangovan[5]

[1]*Mechanical, Amrita Vishwa Vidyapeetham, Tamil Nadu, India*
[2]*Mechanical, Amrita Vishwa Vidyapeetham, Tamil Nadu, India*
[3]*Mechanical, Amrita Vishwa Vidyapeetham, Tamil Nadu, India*
[4]*Mechanical, Amrita Vishwa Vidyapeetham, Tamil Nadu, India*
[5]*Mechanical, Amrita Vishwa Vidyapeetham, Tamil Nadu, India*

Abstract

Injection Molding is a crucial segment of the plastics industry mainly due to its ability and flexibility to manufacture intricate components at high production rates and also with easy process flow. Injection molding has laid its hands over right from small parts to big machine parts. Plastic side release buckles, commonly employed in standard bags, suitcases and pouches, and are manufactured exclusively by the injection molding process all around the world. The process involved should be perfect enough to produce parts without defects and low quality. Gate location is one of the important in this process which is necessary for the filling purposes. In view of the significance of a perfect gate location for the plastic injection mold for the side release buckle, this paper focusses on the complete analysis of the buckle using a moldflow software and the determination of optimum gate locations for it. The analysis is complete look into the various quality features important for the mold, it considers identification and improvement of parameters such as fill time, quality, extent of packing and reduced defects and warpage. Utilization of the optimized gate locations for the mold lead to reduced production costs, higher quality and enhanced competitive power of mold enterprises.

***Keywords**: Injection molding, Side release buckle, Moldflow, Gate location.*

--***--

1. INTRODUCTION

In the present sequence of events, with rapid technological progress and advancement, a new era of miniaturisation, weight reduction and the search for alternative materials. Apart from the improved performance and added benefits, the part complexity and manufacturability also gets elevated. Thus to cope up with these disadvantages, drift towards newer technology in every stage of the product lifecycle is proving to be vital. As this technology is in its relatively early stages of development, further optimisation of its various parameters can lead to even more efficient systems and better performance. One such instance is the use of plastics as an alternative to metals for reduced load carrying operation and for minimising the weight. This coupled with the need for miniaturisation requires improved manufacturing techniques like high precision injection moulding enhanced by utilizing an optimised injection mould design.

The surge in the usage of plastics is mainly due to their lower weight, melting temperature, and cost as compared to other materials like metal, composites etc, accompanied with decent flow characteristics. Thus the demand for smaller, precise and compact designs with intricate geometries, on using plastics, has to be met. Injection moulding is an ideal plastic manufacturing process due to its ability to manufacture complex plastic parts with high precision and production rates at low operation costs with only a relatively high initial investment for mould design and fabrication [1]. Mold flow analysis and design of an optimum injection mould can thus minimise costs, improve filling and cooling parameters, increase packing efficiency and finally reduce the mould defects. This paper is based on the mold flow analysis of a plastic side release buckle with an existing standard design, as shown in Fig.1, in order to determine the best gate/injection locations for its mould. Plastic side release buckles are widely used for carrying moderate loads, specifically in carry bags, suitcases, straps and seatbelts. Analysing and improving its mould design characteristics, thus can be assistance to improve the reliability and simplicity, and reduce the cost of the buckle.

2. MOLDFLOW ANALYSIS

Over the past decade, the field of flow analysis has gained increasing importance in injection molding. Flow analysis provides rational solutions to many of the complex effects that cause problems in the injection molding process. These effects have included warping, molded-in stress, excessive fill pressures, part flashing, and others. The interrelationships between part design and molding process parameters were analysed in order to determine the optimum criteria. Practical experience often is insufficient to identify potential problems and too limited to handle the full range of molding problems that can be addressed by techniques such as flow analysis. Hence, much prototyping and mold "fine-tuning" is necessary before successful molded product results can be achieved. Before the main analysis, a preliminary moulding window analysis was conducted to obtain the recommended values for mould temperature, melt temperature and fill time. The moldflow methodology followed is shown below in Fig.2. Series of trials were conducted for different injection location and the parameters were compared.

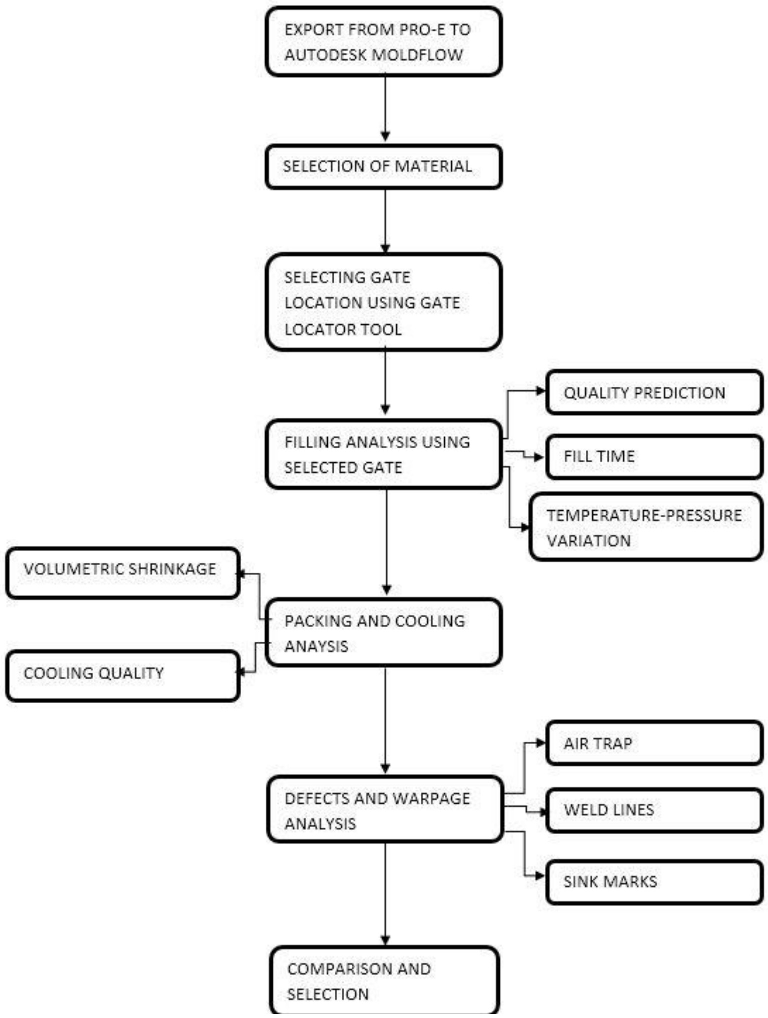
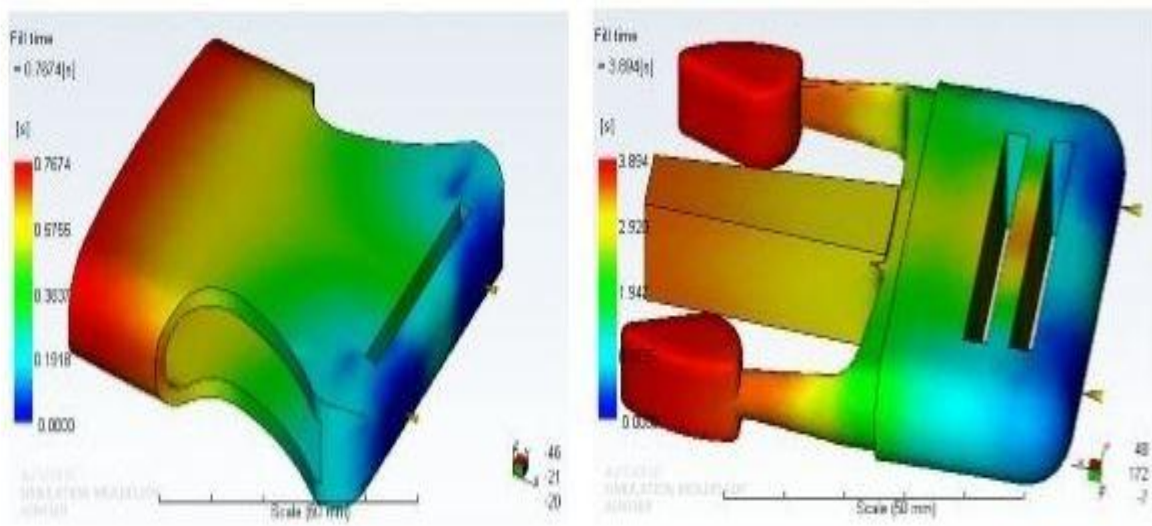


Fig-1: Flow diagram of the analysis process adopted.

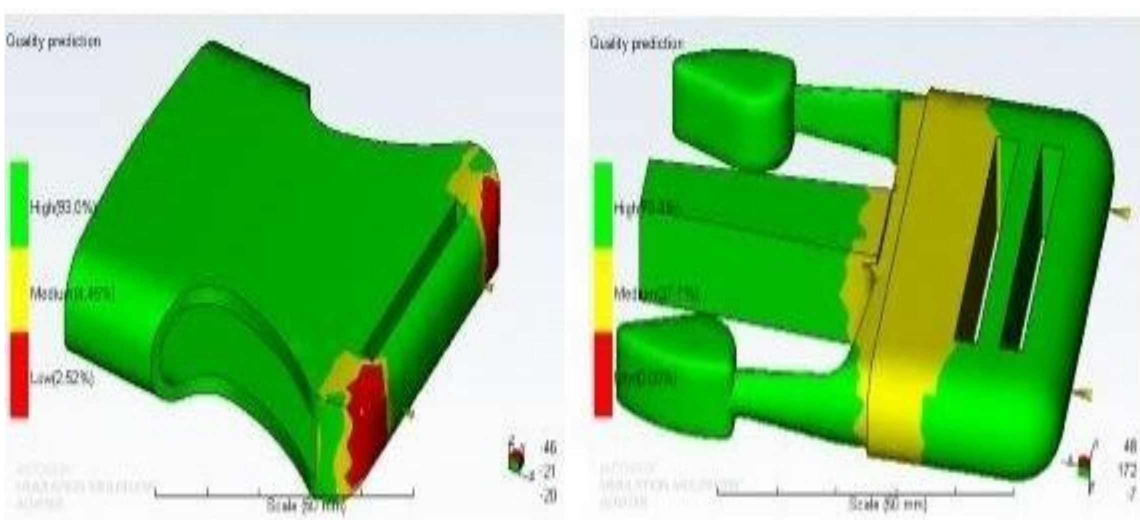
3. FILLING ANALYSIS

There are variable conditions during molding that influence part performance. Of paramount importance are gate location(s) and controlling the cavity fill rate or pattern. The proper fill helps eliminate part warpage, shrinkage, weld lines, and other problems or defects. The filling phase of the injection moulding process greatly affects the quality of the end product. By utilising filling analysis the processing characteristics of an injection mould can be investigated and optimised at the design stage. Filling Analysis simulates how plastics will fill in the mold under given product design, material, machine, mold, gating, and processing conditions. For high productivity and part value, low filling time and high percentage of good quality regions are needed. Also a minimal injection pressure and uniform temperature distribution will help in improved precision. Fig. 2 and Fig. 3 show the filling analysis results for the best gate location. As shown, about 93% of the plug and socket component have high quality and filling times of 0.7 s and 3.8 s are expected for the socket and plug component respectively.



(a) (b)

Fig.2 (a) Fill time of socket (b) Fill time of plug



(a) (b)

Fig. 3: (a) Quality Prediction for socket, (b) Quality Prediction for plug

4. COOLING AND PACKING ANALYSIS

In addition to mold, raw material, and machine costs, the cost of injection molded articles depends on the molding cycle. Mold cooling time is the biggest contributor to the overall cycle time in almost all plastic molding applications (could be as high 70-80% of total cycle time).

Mold cooling analysis will helps optimize cooling layout, reduce cooling time, avoid hot spots, avoid filling/processing issues and most importantly save money [2]. Packing analysis on the other hand determines the net volumetric shrinkage at the end of the process after ejection. This helps determine dimensions to which a cavity and core should be fabricated in order to produce aproduct of desired shape and size. The analysis considers shrinkage as a function of mold temperature, part thickness, injection pressure, and melt temperature. A low shrinkage (<20%) and a moderate cooling quality (>45%) is sufficient for the operational life of the product. As shown in Fig. 4, these requirements are m

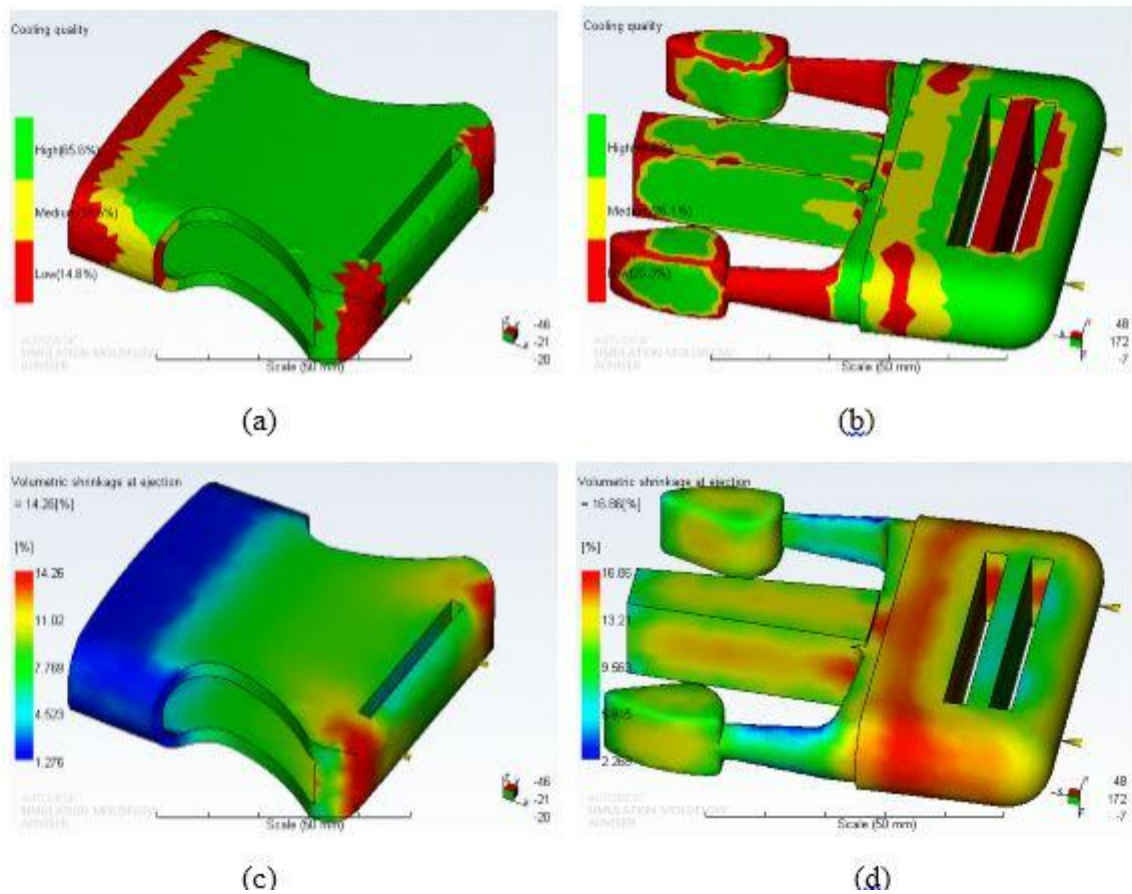


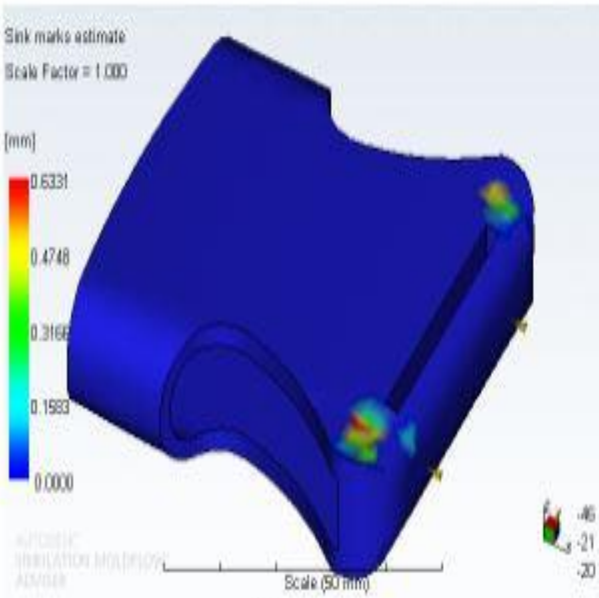
Fig-4: (a) Cooling quality of socket (b) Cooling part of plug (c) Volumetric shrinkage at ejection (d) Volumetric shrinkage at ejection.

5. SINK MARKS AND WARPAGE ANALYSIS

Achieving the dimensions and shape of the target design poses a potential major problem for the production of injection molded plastic products. Owing to the number of complex interactive shrinkages that typically are developed from the molding of the part, it is virtually impossible for even the most skilled designer to take an empirical approach in predicting what their net effect will be on the final molding. Therefore, it is expected that the part cannot be fully evaluated dimensionally or mechanically until a mold is actually built and the part produced. Standard mold filling and cooling analysis can provide the designer with a significant advantage in determining the moldability and optimum conditions required to produce a quality part. However, ultimately the designer wants a quantitative means, rather than an empirical approach, to predict the actual dimension and shape of the part, thus being able to envision how the part will be molded. This is accomplished by the warpage analysis [3]. Apart from warpage the following major and frequently occurring defects must also be estimated and reduced in order to increase the strength and lifetime of the final product:

- Air traps/Voids: Air can be trapped between the melting plastic granules in the injection barrel and be carried all the way into the mold during mold filling.Its pressure causes the dimple. This is likely to cause part ejection problems.
- Weld line: is the line where two flow fronts meet when there is the inability of two or more flow fronts to "knit" together, or "weld", during the moulding process. This can cause a weak area in the part which can cause breakage when the part is under stress.
- Sink Marks: A surface blemish in which the outer wall is depressed in areas of the plastic product. Sink marks are often caused by unwanted voids within the plastic material.

A defect and warpage free product is the ultimate goal of the process. As shown in Fig. 5, the sink marks and warpage is minimum.

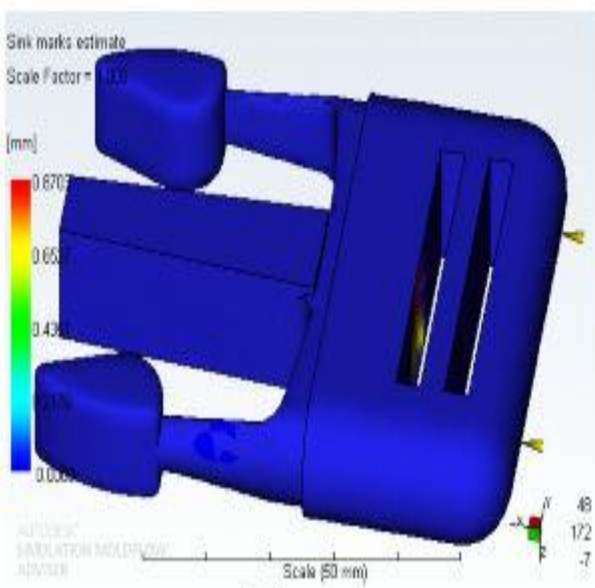


(a)

Property	Numerical Value
Maximum Sink Depth	0.08 (mm)
Average Sink Mark Depth	0.05 (mm)
Percentage of Model prone to Sink Marks	0.34 (%)
Nominal Max Warpage Deflection	1.259 (mm)
Percentage within Max Deflection	100 (%)

(b)

Fig-5: (a) Sink marks estimate of plug (b) Properties with numerical value.



(a)

Property	Numerical Value
Maximum Sink Depth	0.87 (mm)
Average Sink Mark Depth	0.62 (mm)
Percentage of Model prone to Sink Marks	0.51 (%)
Nominal Max Warpage Deflection	2.227 (mm)
Percentage within Max Deflection	100 (%)

(b)

Fig-5: (a) Sink marks estimate of plug (b) Properties with numerical value.

6. SELECTION AND COMPARISON

After a series of trials and comparisons were conducted, the second gate location shown in both the tables below were selected for the plug and socket component [4]. The selected injection locations better gave flow parameters compared to the others. Utilization of this optimized design gives higher productivity and quality.

Table - 1: Different gate locations with their different parameter values of plug.

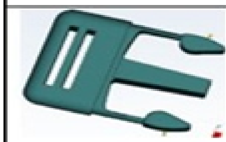
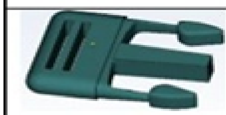
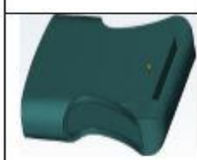
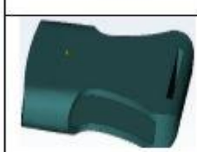
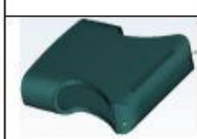
Gate Location	Quality (%)	Fill Time (s)	Cooling Time (s)	Cooling Quality (%)	Volumetric Shrinkage (%)	Sink Mark (%)	Maximum Warpage (mm)
	72.2	4.19	199.29	50.3	17.07	0.63	2.228
	72.3	3.89	201.29	50	17.06	0.51	0.227
	72.4	3.90	203.29	50.3	16.9	0.95	0.228
	72.7	3.90	199.29	50.3	16.9	0.92	0.228

Table - 2: Different gate locations with their different parameter values of socket.

Gate Location	Quality (%)	Fill Time (s)	Cooling Time (s)	Cooling Quality (%)	Shrinkage (%)	Sink Mark (%)	Maximum Warpage (mm)
	93	0.78	25.02	65.7	14.04	0.31	1.26
	93.1	0.77	24.76	66.0	14.09	0.34	1.259
	93	0.80	24.94	66.2	14.0	0.34	1.259
	93	0.70	24.79	65.8	14.80	0.33	1.26

7. CONCLUSIONS

Gate location is an important and critical area in plastic injection molding and in die designing. An optimum gate location is always the desired by the mold maker while making a mold for any component. Gate position and the best one from the available is always found or taken into account based on the mold maker's experience.

Above results are of different analysis starting with a preliminary molding window analysis, then a filling analysis, cooling analysis, packing analysis and warping analysis for a quick release buckle. During this variation in their performance based on different trials with single gate and multi gate were made. The results clearly describes the optimum gate position for different scenarios based on which a mold can be fabricated and a gate can be fixed. From the above results parameters like mold temperature, melt temperature, fill time, quality prediction, cooling time, cool quality, volumetric shrinkage, injection pressure and temperature and extent of warpage were successfully determined and later the gate position which shows the closest values to these parameters was taken into account and finally fixed as optimum gate position.

Thus with the help of mold flow software, it was able to predict a gate location keeping all parameters in mind for a small component like quick release buckle and hence use of such types of tools for analysis is proven to be helpful.

REFERENCES

[1] Castro, C. E., Rios, M. C., B., L., & M., C. J. (2005). Simultaneous Optimization of Mold Design and Processing Conditions in Injection Molding. Journal of Polymer Engineering, 25(6), 459-486

[2] Chang, T. C., & Faison, E. (2001). Shrinkage behavior and optimization of injection molded parts studied by the taguchi method. Polymer Engineering and Science, 41(5), 703-710

[3] Huang, M. C., & Tai, C. C. (2001). The effective factors in the warpage problem of an injection-molded part with a thin shell feature. Journal of Materials Processing Technology, 110(1), 1-9

[4] Oktem, H., Erzurumlu, T., & Uzman, I. (2007). Application of Taguchi optimization technique in determining plastic injection molding process parameters for a thin-shell part. Materials and Design, 28(4), 1271-1278

BIOGRAPHIES

H. Adithya Bhat, B.tech, Mechanical Engineering

Sandeep Subramaniam, B.tech, Mechanical Engineering

Arvind K. Pillai, B.tech, Mechanical Engineering

Edu Krishnan L, B.tech, Mechanical Engineering

Dr. M. Elangovan, Professor, Mechanical Engineering