International Journal of Advance Research in Engineering, Science & Technology(IJAREST),
ISSN(O):2393-9877, ISSN(P): 2394-2444,
Volume 2,Issue 9, September - 2015, Impact Factor: 2.125

Analysis of rubber bush mould Using Mould-Flow and compare with experimental result

***H. H. Makwana*[1]**

[1]*Mechaniccal, A.I.T.S., hiren_sos@yahoo.co.in*

Abstract

This With increasingly short life span on consumer electronic products such as mobile phones are becoming more fashionable, injection molding remains- the most popular method for producing the associated plastic parts. The process requires a molten polymer being injected into a cavity inside a mold, which is cooled and the part ejected. The main phases in an injection molding process therefore involve filling, cooling and ejection. The cost-efficiency of the process is dependent on the time spent in the molding cycle. Die design is the most time consuming stage, so here mouldflow is used to analyze die design and for testing mouldflow result, comparison have been done with experimental design..

***Keywords**-Injection molding; die design; moldflow; analysis; defects.*

INTRODUCTION

Injection moulding is a manufacturing technique for making parts from both thermoplastic and thermosetting plastic materials in production. Molten plastic is injected at high pressure into a mould, which is the inverse of the product's shape. After a product is designed, usually by an industrial designer or an engineer, moulds are made by a mould maker (or toolmaker) from metal, usually either steel or aluminum, and precision machined to form the features of the desired part. Injection moulding is widely used for manufacturing a variety of parts, from the smallest component to entire body panels of cars. Injection moulding is the most common method of production, with some commonly made items including bottle caps and outdoor furniture.

DIE DESIGN

Mould a hollow form or cavity into which molten plastic is forced to give the shape of the required component. The term generally refers to the whole assembly of parts that make up the section of the moulding equipment in which the parts are formed. Also called as a tool or die.

Existing design

According to industrial design a transparent view of existing design is shown

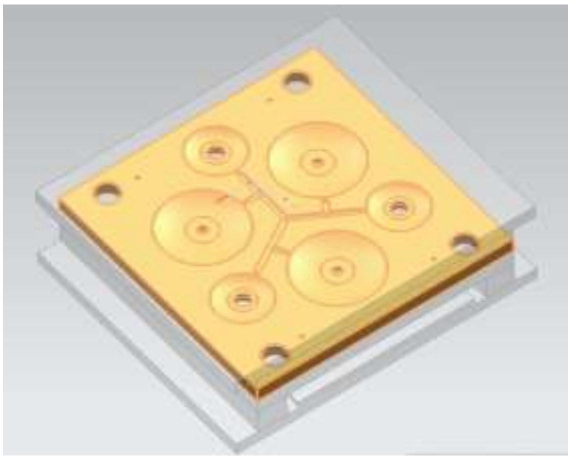

Figure 1. Transparent view of die

ANALYSIS

For Analysis of the mould, MOULDFLOW is used with the required solution criterion like nominal thickness, draft angle, undercut, gate location, cooling time, weld lines, sink marks etc.
Result taken of different effects using mold flow is broadly describe bellow

Analysis of sink marks

- Sink marks is the depression in a moulded part caused by shrinking or collapsing of the resin during cooling which is completely eliminate in the new mould which is shown in figure 7.28
- Max sink depth = 1.12mm
- Average sink mark depth = 0.05mm
- Percentage of model prone to sink marks = 0.58%

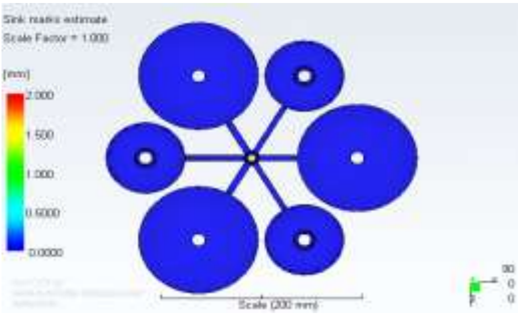

Figure 2. Sink mark

Analysis of warpage

- Here in existing design the main problem is warpage that is eliminated in the new mould which can be shown in figure 7.32 the deflection is only 2.145mm.

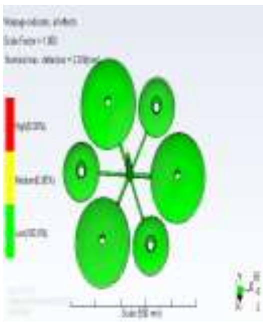

Figure 3. Warpage for differential shrinkage

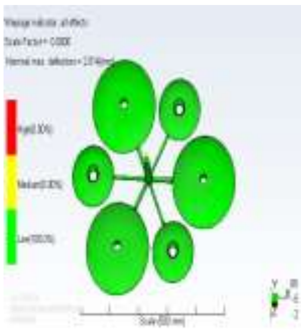

Figure 4. Orientation effects due to warpage

- Due to shrinkage what will be the warpage value is also shown in figure 7.34 in which deflection is to be occurred by 2.014mm.

Analysis for best gate location:

- With this analysis we get the exact idea whether our gate location are appropriate or not.
- In figure 5 the existing gate location is to be shown from that the coordinate of the existing gate location is
  7.77, 18.16, 54.45
- But the existing gate location is not properly located we have to change the gate location as per software suggestion gate location should be the
  18.97, -1.84, 64.37

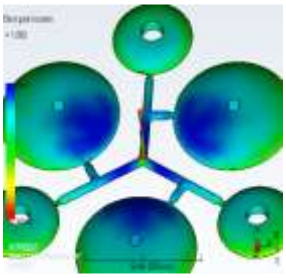

Figure 5. Best gate location

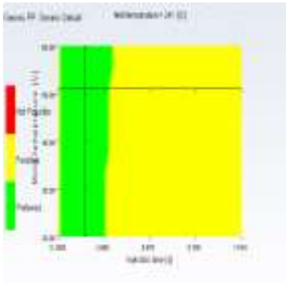

Figure 6. Melt temperature

- From analysis we can get the optimum mould as well as melt temperature
- Optimum point: Mould temperature: 51.1 (C); Melt temperature: 241.1 (C);
- Injection time: 1.75 (s)

6.4 Analysis for weld lines and fill time:

- Here, in part weld line is to be occurred nearer to the inner edge as shown in figure 6.6
- From figure 7 we can identify the how much time is required to reach ejection temperature i.e. 96.95 sec. also from figure 8 we got the temperature value at flow front i.e. 240^0C.
- By analyzing figure 11, we exact idea about to when part will be filling. Here as shown in figure the small part taking more time to fill compare to the bigger one.
- Pressure drop in the part cavity is to be shown in figure 10, quality of the part will be also predict by the mould flow which can be describe in figure 6.9.

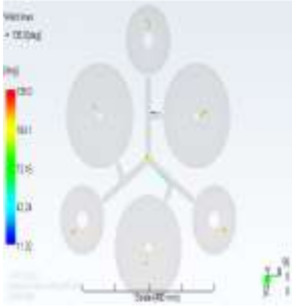

Figure 7. Weld lines

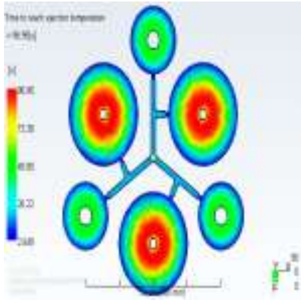

Figure 8.Time to reach ejection temperature

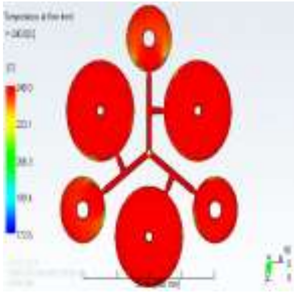

Figure 9.temperature at flow front

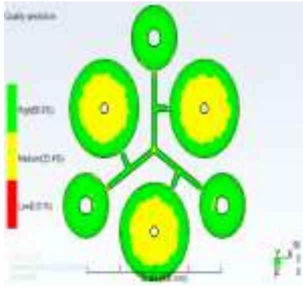

Figure 10. Quality prediction

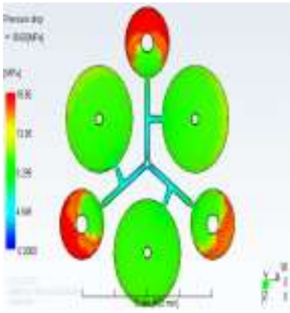

Figure 11. Pressure drop

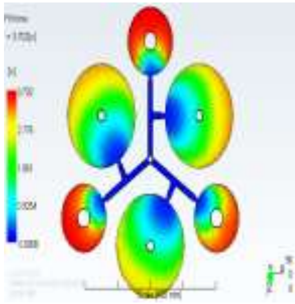

Figure 12. Fill time

- Due to stresses generated in the part what affect it into the skin is shown in figure 6.12
- Maximum Injection pressure is 18.13MPa, changes in pressure in the parts are shown in figure 6.13 sometimes due to variation in pressure some air are trapped during filling which is shown in figure 6.17.

RESULTS AND CONCLUSION

Here, in table 1 the comparison between experimental result and analysis result has been carried out.
According to that almost all data of MOULDFLOW are relevant. It gives the Analysis result satisfactory.

Sr. No.	Property	Experimental result	Analysis result
1.	**sink mark**		
	Max sink depth(mm)	1.10	1.12
	Average sink mark depth	0.15	0.05
	Percentage of model prone to sink marks	Less than 1%	0.58%
2.	**Warpage**		
	Nominal max. Deflection(mm)	1.85	2.01
	Percentage exceeding Nominal max. Deflection	0.0	0.0
3.	**Density(gm/cm^3)**	0.9	0.9
4.	**Shear stress**	0.530	0.534
5.	**Melting point(^{0}C)**	240	240

Table 1. Comparison of experimental result to the analysis result

REFERENCES

[1] Jay Shoemaker, “Moldflow Design Guide: A resource For Plastics Engineers”, 2006, 141-200

[2] Sanjay K Nayak, Pratap Chand.a Padhi, Y. Hidayathullah, “Funadamentals of Plastics Mould Design”, 2012, 2.1-2.124

[3] Yi Yang, Furong Gao, “Adaptive control of the filling velocity of thermoplastics injection moulding”, Control Engineering Practice 8, 2000, 1285-1296.

[4] M.W. Fu, J.Y.H. Fuh, A.Y.C. Nee , “Undercut feature recognition in an injection mould design system”,Computer-Aided Design 31, 1999, 777–790