International Research Journal of Engineering and Technology (IRJET) e-ISSN: 2395-0056
Volume: 07 Issue: 02 | Feb 2020 www.irjet.net p-ISSN: 2395-0072

INJECTION MOLDING METHODS DESIGN, OPTIMIZATION AND SIMULATION OF PLASTIC CAP BY MOLD FLOW ANALYSIS

Prashant paraye[1], Aman kumar[2], Atul kumar tiwari[3]

[1]*Engineering Mechanical department, CIPET Bhopal*
[2,3]*Student, CIPET Bhopal*

---***---

Abstract -*Mold flow simulation helps designers to see how their designs will be resulted after injection molding process without needing to do the Injection Molding process. The use of simulation programs saves time and reduces the costs of the Molding system design. Injection molding design simulation holds an important role in analyzing the outcome of the design. In this paper plastic cap part is analyzed and studied to solve the problems frequent rejections due to as shrinkage, weld lines, air traps, and sink marks. All the designs were simulated with Autodesk Mold flow Adviser. Today, many manufactures have proven mold flow analysis (MFA) to be the medium between a flawless design and production. This not only improves the quality but also help us to guide about the selection of machines and the production planning.*

Key Words: Injection moulding, Mould design, Mold flow simulation, Optimization Plastic Injection mould, Mould Flow Plastic Advisor (MPA)

1. INTRODUCTION: Injection Moulding is one of the common methods to do the mass-production of plastic product. The Autodesk Simulation Moldflow results help to identify the main problem areas before the part is manufactured that are particularly difficult to predict with traditional methods. Analysis is essential for designing and mould making through simulation step-up and result interpretation to show how changes to wall thickness, gate location, material and geometry affects manufacturability. On the study of injection moulding process the most important point that lies under is the mould. The mould, which is the most important component part, that gives the product the shape required and designing the part product associated with the mould, hence worth study. In injection moulding the material is allowed to cool, the mold is opened, and the solid product inside is ejected into a collection hopper. Common problems associated with injection molding are numerous.

2. OBJECTIVES OF THE WORK:

1. To study the component design and identify the critical parameters in the molding window analysis.

2. To set optimum process parameters like injection pressure, speed, and temperature

3. To achieve the minimum production cycle time

4. To foresee the possible problem for a product design; and therefore able to op-timize the design in the mould design process.

5. To construct a rapid prototyping of the mould cavity design into a standard mould plate.

3. PROBLEM DEFINITION:

2. Identify the critical parameters in the molding window analysis. (Material Selection, Experimental setup for injection molding machine, determining the molding conditions.)

3. To set optimum process parameters setting. (Design of Experiments using minitab and Trials on Injection molding machine)

4. To simulate the flow and finding fill time, injection time and molding defects.

5. Validation of results by comparing the moldflow results and trials.

4. Model details:

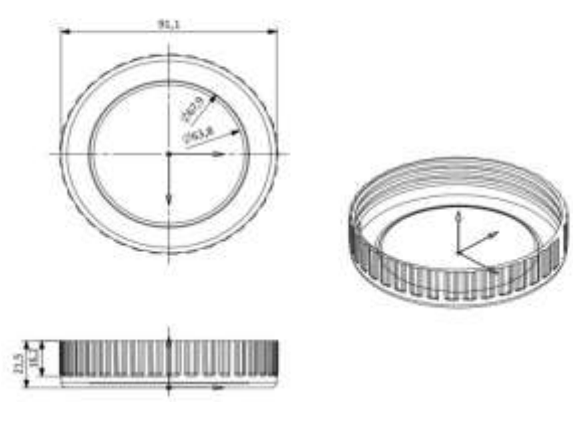

Fig.1.a) CAD Model of plastic cap

5. Process setting:

Melt temperature: 210 (c)

Mold temperature: 60 (c)

Max. machine injection pressure: 180(MPa)

6. Material Data:

	Family name- PP Trade name- Repol H110MA
Modulus of Elasticity	1340 MPa
Poison ratio	0.392
Shear modulus	481.3 MPa
Shrinkage value	0.7-1.5%

Fig:1.b) product

7. Simulation result:

7. A) Fill analysis result:

The Fill time result shows the position of the flow front at regular intervals as the cavity fills. At the start of injection, the result is dark blue, and the last places to fill are red. If the part is a short shot, the section which did not fill has no colour. Fill time is the time taken to fill up the part inside the cavity; it is also to show how the plastic material flows to fill the cavity. Which takes for all full filling of the cavity to be duration of 1.569 s.

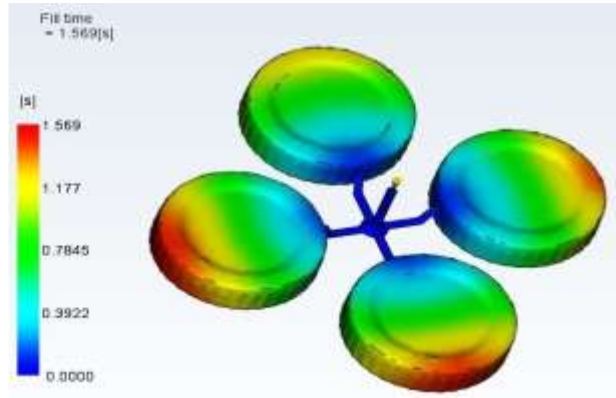

Fig.7A) Fill time analysis result

7. B) Confidence to fill analysis result:

It displays the probability of a region within the cavity filling with plastic at conventional injection molding conditions. It shows the figure that confidences of fill of complete part are high. If the cavity does not fill (short shot), the changes must be made to the processing conditions, injection locations, design of the parts, or choice of the plastic. Table shows the risk of the part filling base on colors.

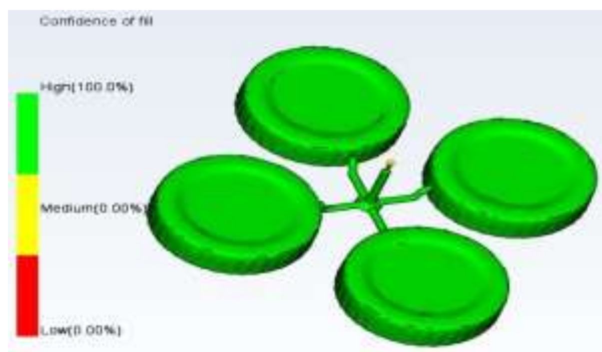

Fig.7B) Confidence to fill result

7. C) Injection pressure analysis result

It is produced by a Fill analysis, shows the maximum injection pressure value obtained before the velocity/pressure switch-over occurs during the filling phase. The result obtained from the simulation of the analysis shows that though a maximum pressure in the red zone is allowed as high as 17.82 MPa the melt can completely fill the cavity with the yellow region of maximum optimum injection pressure of 13.36 MPa that would be enough for complete filling, and hence this pressure is enough for the injection pressure of the mould.

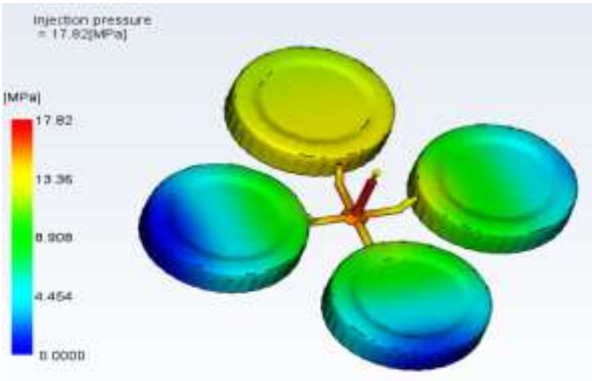

Fig.7C) injection pressure analysis result

7. D) Flow front temperature analysis result:

If the flow front temperature is too low in a thin area of the part, hesitation or short shot may be occurred. If it is too low in an area where weld lines are present, the weld lines may appear worse.

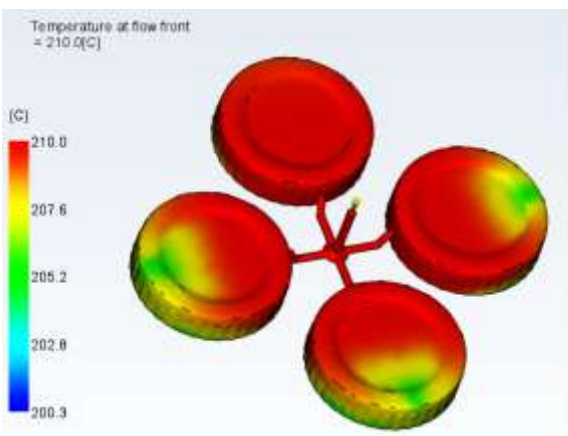

Fig.7D) flow front temp. analysis result

7. E) Pressure drop

The Pressure drop result uses a range of colors to indicate the region of highest pressure drop through to the region of lowest pressure drop. This result indicates how much pressure is necessary to fill the different areas of the part.

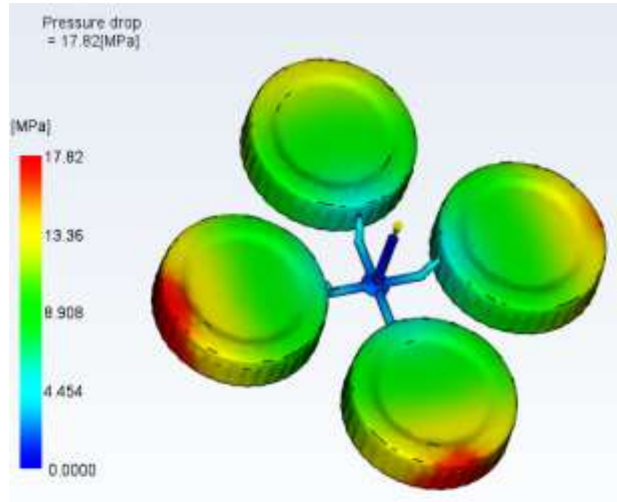

Fig.7E) Pressure drop analysis result

7. F) AIR TRAP Analysis result

Air trap results shows the flow pattern with respect to time during the filling process. Air trap result shows the possible locations where the air trap could occur. An air trap is formed by converging melt fronts trapping a small bubble of air.

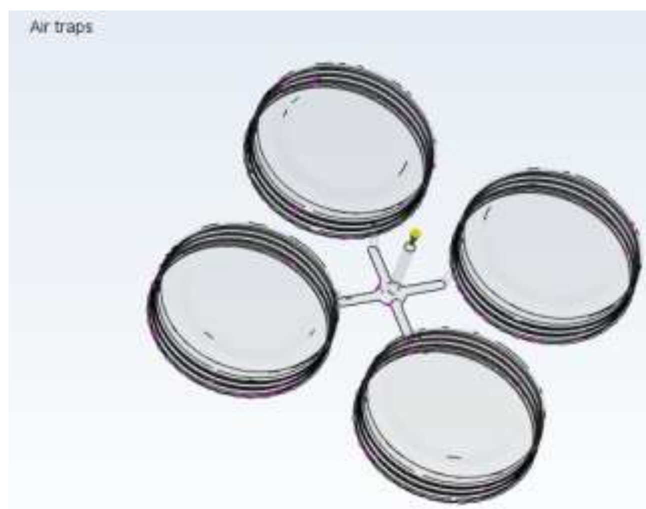

Fig.7F) Air trap analysis result

7. G) Weld line analysis result

Fig. 7G) Weld line analysis result

7. H) Cooling quality analysis result

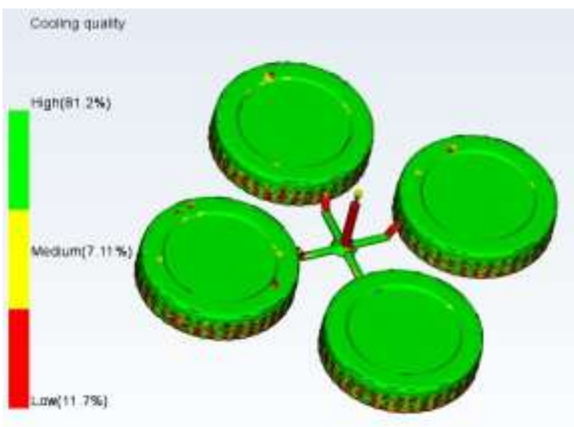

Fig. 7H) Cooling quality analysis result

8. RESULTS AND DISCUSSION

Table - 1: Cooling analysis result table of cap

Cooling quality	
Maximum temperature variance	5.5 (C)
Minimum temperature variance	-6.8 (C)
Maximum cooling time variance	28.00 (s)
Minimum cooling time variance	-2.75 (s)

Table – 2: Comparison with Mold Flow analysis Result

Result	OLD PROCESS PARAMETERS	NEW PROCESS PARAMETERS (mould flow)
Material trade name	Repol H110MA	Repol H110MA
Melt temperature	300	210
Mold temperature	40	60
Injection Locations per cavity	1	1
Actual fill time	2.80 (s)	1.56 (s)
Act. Injection pressure	45 MPa	17.82 MPa
Flow front temp.	220	210
Pressure drop	21.5 MPa	17.82 MPa
Clamp force area	289.5 (cm^2)	313.86 (cm^2)
Max. clamp force during filling	65 (tone)	29.39 (tone)
Vel./pressure switch-over at time	3.0 (s)	1.51 (s)
Estimated cycle time	35.0 (s)	19.3 (s)

This analysis can help predict short shots. Short shots are a legitimate concern for those involved in creating plastic parts. If you have a component with variable wall thickness, it is important to run an analysis to make sure these areas will fill out. By observing the analysis results, The confidence of fill is high when One Gate Location is taken other defects like air traps, weld lines will be less.. Cooling Time and Fill time were two response variables. Total nine experiments were conducted and S/N ratios for cooling time and fill time were plotted. Confirmation tests were conducted using the higher the better values for cooling time and fill time. After making the prototype simulation with mold flow, it can be concluded that it makes the ground for the manufacturing of Plastic cap part practically in process for injection molding.

10. References

1. M. Salunke, "Injection molding methods design, analysis and Simulation of plastic cup by mold flow analysis", international journal of current engineering and scientific research (IJCESR), ISSN (print): 2393-8374, (online): 2394-0697, volume4, issue-6, 2017.

2. K. Mesfin, "Product design, simulation and analysis of manual juice maker",Identification Number 10328, Degree Thesis PTE, 2014. 22. J. Shoemaker, "Moldflow Design Guide,A Resource for Plastics Engineers", Moldflow Corporation,Moldflow Design Guide, First Edition, Framingham, Massachusetts, U.S.A [Text Book].

3. Peter Kennedy, Rong Zheng, Flow Analysis of Injection Molds, 2nd Ed.

4. http://www.engineersedge.com/injection_molding,.htm, 2006

5.http://www.efunda.com/DesignStandards/plastic_design/plastic_intro.cfm, 2006