Research Paper Engineering

Design Enhancement for A Plastic Injection Mold for An Automotive Component Using Results of Simulation for Input

* Mr. Kiran C. Mahale **Prof. Avinash. V. Patil

*(M.E Design Engg.) SSGBCOET, Bhusawal.

** HOD Mechanical, SSGBCOET, Bhusawal

ABSTRACT

Mold Design forms the basis of the development work required for producing the desired number of units in a given time frame. Multi-cavity molds can help produce the components in mass quantities in a short duration. The simplicity of the mold is the key to ensure the quality of the component produced and the associated costs of development. A review of the same with ingenious inputs in the design phase would help the Company to achieve its overall objectives. As such, the objective of this dissertation work is to Design a Plastic Injection Molded Component for the Automotive Industry that would ensure a minimal time for development of the mold as well as deliver a good quality product during trial and testing. This would in turn help the overall product development process at the subassembly and the final assembly level of the Product development cyc

Keywords : mold, injection, design , molding

1. Introduction

Mold Design forms the basis of the development work required for producing the desired number of components in a given time frame. Multi-cavity molds can help to produce the components in mass quantities in a short duration. The simplicity of the mold is the key to ensure the quality of the component produced and the associated costs of development.

A review of the same with ingenious inputs in the design phase would help to achieve its overall objectives.

The molding may cause defects and its processing offers a challenge during its development phase. The cost of the mold is high and any process that is not optimized renders heavy overheads during its development cycle and production. So designing the mold which ensures best suitability for the features on the component with smooth flow of molten plastic is very important part of development process.

The successful launch of any plastic product depends on knowing the true costs and profitability before the job is started. Injection molding typically involves large volumes of parts. Small cost overheads per part can be compounded to large cost differences over the life span of the part. Major cost components considered here are material, re-grind and machine costs. Scrap, rejections and regrind costs are also accounted in the cost.

2. Problem Definition

A large automotive ancillary industry in Pune has offered AbleTech to Design the mold of `Casing' for a two-wheeler switch and `Housing' for a premium four-wheeler. The Design of the Mold is to be accomplished upon checking the manufacturing feasibility while making the development cost-effective.

The molding may cause defects and its processing offers a challenge during its development phase. The cost of the mold is high and any process that is not optimized renders heavy overheads during its development cycle and production.

- Design for a Plastic Injection Mold required more time.
- Practical purpose Trial & Testing more time is required.
- To study the component design with the prospective of a mold designer.
- Generate a layout using 3D modeling tools.

3. Methodology

The following steps are involved for achieving the objectives of the project that can be enumerated as

- Generation of 3D model of component from 2D drawing.
- Study the component design with the perspective of a Mold Designer.

(Query/ Analysis report)

- Identification of the critical features that would call for special elements while designing the mold, such as critical dimensions, tolerances, surface finish, abrupt changes in thickness, undercuts.
- Generation of a rough layout for the mold design. Generate a layout using 3D modeling tools (3D model generation by using tools like CATIA-V5/ UG)
- Design and detailing for manufacturing the mold. (Final 3D model and 2D drawings)
- Design a simplified mold as per the functional requirements of the component.
- Review and get the approval from the product designer for changes in design.
- Design validation of Mold for cycle time optimization and required level of dimensional accuracy, surface finish and strength.

i. Experimentation

The draft analysis and/or the flow analysis of the component would provide useful inputs for anticipating the performance of the component during its processing phase. It is generally not feasible to generate a soft mold for experimentation because of high cost involved .Experimentation of mold design will be done by varying the parameters like type of gate, gating system location, venting location and location of runners and risers for producing the defect free component. These parameters will be changed at least in three levels and appropriate experimentation method will be followed.

From the simulation and analysis, Mold flow software provides sufficient information regarding its filling time, injection pressure and pressure drop. With these results, users can avoid the defect of the plastic in actual injection such as sink mark, hesitation, air traps, and over packing. The analysis will also help the mould designer to design a perfect mold with minimum modifications and which will also reduce the mold setup time. With this analysis and simulation, it will help to reduce time and cost.

ii Validation

The design will be validated by producing the component with the help of the developed mold without affecting the component's functionality. Flow of plastic will be observed. Dimensional accuracy will be measured and checked with the specified dimensions. Visual and actual inspection will be done while attempting to identify the defects. Further, the component will be checked for fitment in the sub-assembly.

4. Software Analysis

i. Fill Time:-

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 mould. From that we know that the short shot (part of the model which did not fill) part will be displayed. From that result one can also understand how the weld line and air trap will form. Figure1 shows the material filling into the mould.

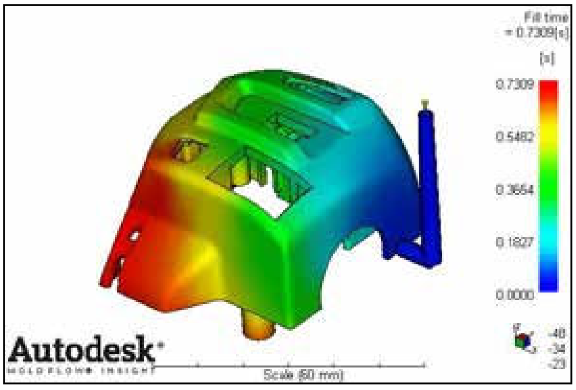

Figure 1: Study without Rib: Fill Time

ii. Injection Pressure

The colour at each place on the model represents the pressure at the place on the model. Two colures show the highest pressure (red) and lowest pressure (blue). The injection pressure can be used in conjunction with pressure drop result. For example, even if a section of a part has an acceptable pressure drop, the actual injection pressure in the same area may be too high. High injection can cause over packing.

To reduce the chance of this happening, follow these steps:

1. Increase the maximum injection pressure
2. Alter the polymer injection location
3. Alter part geometry
4. Select a different material

Figure 2 shows the result of injection pressure 44.97 MPa.

iii. Pressure Drop:-

Figure 2 shows the pressure drop result. When the place is filled, the pressure will drop from the injection location to the end of the filled part. The pressure drops result uses a range of colors to indicate the region of highest-pressure drop (colored red) through to the region of lowest pressure drop (coloured blue). The pressure drop is one factor used to determine the confidence of fill result

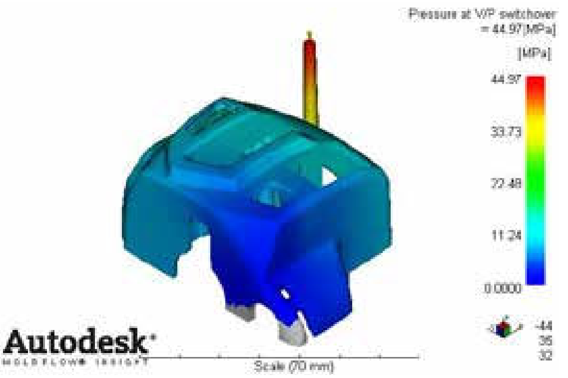

Figure 2: Pressure at V/P Switchover

iv. Air Traps:-

Figure 3 shows the air traps result after analysis. The small blue bubbles are showing the air traps in the parts. Air traps result shows the regions where the melt stops at a convergence of at least 2 flow fronts or at the last point of fill, where a bubble of air becomes trapped. To prevent air traps occurring when converging flow fronts surround and trap a bubble of air, balance flow paths by either:

1. Using flow leaders / deflectors
2. Changing part wall thickness
3. Changing polymer injection locations

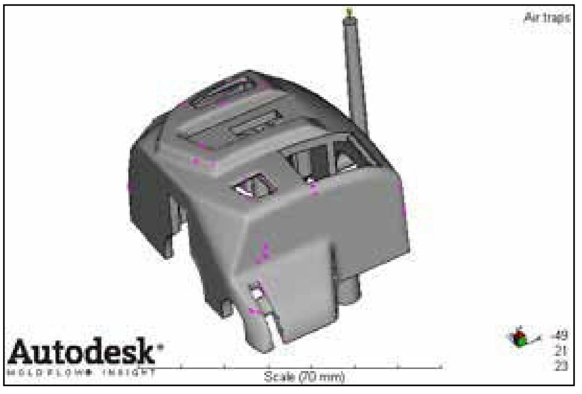

Figure 3: Rib: Air Traps

V. Setting Hold Time:-

Knowing whether the part should be run with the gate sealed or unsealed allows you to fix the length of time for the second stage. If gate-seal time is required, add a second or two to the gate-seal time for process robustness and stability. This does not have to add to cycle time, as most situations allow you to take this off the cooling time or mold-closed timer. If gate-unsealed is best for part performance, then start with half the time it takes to get the gate frozen.

Picking the exact gate-seal time is the worst possible case due to normal temperature and process variations. Your parts will be made sometimes *with* and sometimes *without* gate seal. This will produce inconsistent parts. A correlary is that if your process runs without gate seal, cycle-time consistency becomes critical for consistent parts. If cycle time varies while running with gate unsealed, your parts will vary because you are changing the amount of polymer in the cavity. You can check this by weighing parts.

Vi. Setting Hold Pressure:-

Finding the correct hold pressure is critical for packing the part. The right second-stage pressure centers the process within the range of part specifications to attain a good Cpk for your parts. Since hold time is set from the gate-seal experiment, you should experiment to find the correct second-stage pack and hold pressure to center the part within its specification range.

First, check that the first stage is stable and that you have the expected short or sinky part at the end of the first stage. Do this by leaving time on the second stage and reducing hold pressure to a slow as the machine allows—but do not take hold or pack time to zero. Is the first stage as expected? If not, fix the first stage. Nothing you can do in the second stage will compensate for a fouled-up first stage.

If the first stage is stable, start adding hold pressure. Start low, perhaps with only 1000 to 2000 psi plastic pressure. Check the parts with each increase in hold pressure; keep raising hold pressure in small increments until your best judgment indicates these parts might be acceptable. Make the quantity of parts that QC needs for preliminary testing. Label and set them aside First, check that the first stage is stable and that you have the expected short or sinky part at the end of the first stage. Do this by leaving time on the second stage and reducing hold pressure to a slow as the machine allows but do not take hold or pack time to zero. Is the first stage as expected? If not, fix the first stage. Nothing you can do in the second stage will compensate for a fouled-up first stage.

If the first stage is stable, start adding hold pressure. Start low, perhaps with only 1000 to 2000 psi plastic pressure. Check the parts with each increase in hold pressure; keep raising hold pressure in small increments until your best judgment indicates these parts might be acceptable. Make the quantity of parts that QC needs for preliminary testing. Label assnd set them aside

5. Result and Discussion

- **Study Without Rib: Fill Times :-**

The fill time result is shown in the fig. And the component filling time is 0.73s & Red colored portion on the part indicates that the area fills at last figure 1.

- **Pressure at V/P Switchover :-**

The Pressure at V/P switchover result shown in the figure 2.

- **Without Rib: Air Traps :-**

Air traps result shown in the above figure 3.

Table 1: Compression Analysis Result of Moldflow & Physical Experimental result

Sr. No	Parameters	Notation	Analysis of Results Moldflow	Physical Experimental Result
1	Part Overall Size		75mm X 55mm X16mm	75mm X 55mm X16mm
2	Thickness		Min = 1mm, Max= 2mm	Min = 1mm, Max= 2mm
3	Melt Temperature	(◦) C	290	260
4	Injection Pressure	(MPa)	44.97	60
5	Holding Pressure	(MPa)	40	42
6	Total Cooling Time	(Sec}	12	18

6. Conclusion

- The Design of the Mold and the processing parameters has an influence over the quality of the component produced.
- Defects can be minimized through improved design of the mold with the study of simulation of flow through the mold.
- The material, size, intricacy (complexity) and the rate of production required should be studied for evolving the right Mold design for the given component.

7. Future Scope

Software like `Mold Advisor' or `Moldex' can also be employed to compare results of the analysis with the software used for study here.

- Complexity of the component can be linked to the analysis for richer evaluation
- The analysis could be done for alternative Engineering materials for the given application of the component

REFERENCES

[1] Plastic Injection Molding Process"Pertanika J. Sci. & Techno. Supplement (2004) [85-99]. | [2] Wong, C. T., Shamsuddin Sulaiman, Napsiah Ismail & A.M.S. Hamouda Design and Simulation of Plastic Injection Moulding ProcessPertanika J. Sci. & Techno!. Supplement 12(2): 85 - 99 (2004). | [3] Fauzun, M. Hamdi, A.E. Tontowi, T. Ariga, "Formulation of the size and position of spiral cooling channel in plastic injection mold based on fluent simulation results". IEEE 2008 [1728-1733] | [4] R. Dubay, B. Pramujati and J. Hernandez, "Cavity temperature control in plastic injection 1 C. T. Wong,Shamsuddin-Sulaiman, Napsiah Ismail & A.M.S. Hamouda, "Design and Simulation of molding, IEEE July 2005 [911-916]. | [5] E O'Neill, C. Wilson and D. Brown, "The benefits of solid modeling in the plastics injection molding industry", IEEE April 1997 [99-107]. | [6] M. Dastagiri, M. Prasad and M Annamacahrya , "Application of Axiomatic Design in injection molding process," IEEE 2010 [222-228]. | [7] Chi-Huang Lu andChing-Chih Tsai,"Multivariable Self-Tuning Temperature Control for Plastic I