Design Development and Analysis of Manifold and Coupler Body of Air Inflator Product

Prashant.V.H[1] and Ramesh BabuK[1]

[1] Dept. of PG studies, Govt. Tool Room & Training centre, Mysore-570016
Email: pvhilalpure@gmail.com, rameshbabumys@gmail.com,
Ph: 8792220238, 9141629597

ABSTRACT

Injection molding is a most common and important of all plastic processing process. This process is extremely versatile, and can produce very complex shaped parts, with the use of multi-sided molds. This paper deals with the design development and mold flow analysis of injection mold tool for two components of same assembly and are of different materials (i.e., manifold and coupler body). The part was analyzed with high end analysis software and accordingly the design has been carried out. The tool has been manufactured after some trials and corrections. The mould flow analysis software is being used to predict the parameters like runner location, fill time and weld lines etc before production of the tool. Any product to be manufactured invariably requires machine and tool. Tool design and development is specialized and critical area, the tool design should match the machine specification and should be accurate and economical for good product.

Key words- Injection mold design, Weld lines, Fill time, Air traps, mold flow.

1. Introduction

Injection molding is a method of processing predominantly used for thermoplastic polymers. Thermoplastic material processed initially from melting the material, then forcing the molten material into a steel mold, where it cools and solidifies. Solidified product is ejected by opening the two halves of mold and ejected by ejector assembly[1]. It is the one of the principal tool in producing the highly complicated and elegant products.

Injection molding is a versatile process allows us to produce high quality, simple or complex components on a fully automated basis at high speed with materials that have changed the face of manufacturing technology [2].

It involves a process cycle which includes three stages such as fill stage, pack stage and hold stage[3]. Each stage has its own imporantance in the process fill stage involves, the mold cavities are filled with the molten material as the material is forced forward when the material is feed in the barrel which causes mixing. This stage is determined by an injection velocity , a pressure and a time. Next stage involves, as the melt enters the mold it fills the each and every corners of the gap or the product to be formed. Filled molten material is gets solidified with the several cooling applications and introduces or causes several defects like shrinkage, warpage etc., this stage is necessary to be focrce the molten material into the mold and to be form the molded component effectively and precisely. Hold stage involves the molten material which is fed into the mold to be get freezed and ready for the ejection of the product produced. The hold pressure applies forces against the material in the cavity until the gate freezes to prevent leaking of melt.[4,5]

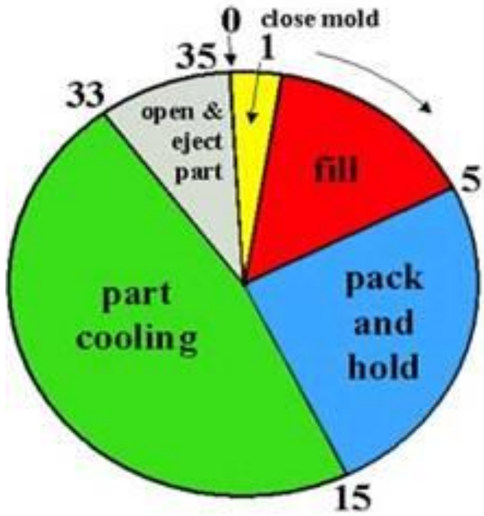


Fig 1: Injection molding cycle

2. Material and Methodology

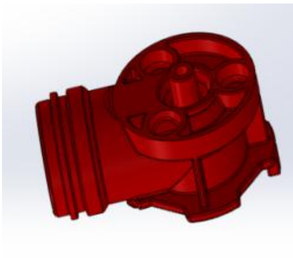


Fig 2: Manifold

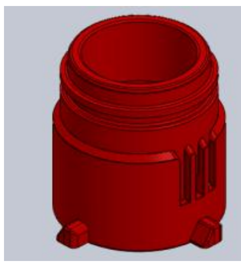


Fig 3:Coupler body

For the component-manifold, Polyoxymethylene (POM) is used for the production of this component. POM-BASF-ULTRAFORM-H2320 006 UNC Q600 is a POM with high molecular weight grade for injection molding. Normally it is used for the thick walled parts.

Density -1.40 g/cm^3
Molding shrinkage - 2.1%

For the component-coupler body, PA6-GF15 (Polyamide-6 Glass reiinforced) Glass fibre reinforced injection moulding grade e.g. for internal gas pressure applications such as steering column switch, automobile mirror housings and wheels of mountain bikes.

Density -1.21 g/cm^3

Molding shrinkage - 0.4-0.6 %

3. Analysis

To avoid the high costs and time delays associated with problems discovered at the start of manufacturing, it is necessary to consider the combined effects of part geometry, material selection, mould design and processing conditions on the manufacturability of a part. Using predictive analysis tools to simulate the injection moulding process, organizations and industries can evaluate and optimize interactions among these variables during the design phases of a project before production begins, where the cost of change is minimal and the impact of the change is greatest.

The productive analysis can simulate the filling, packing and cooling phases of thermoplastics moulding processes using materials with or without fillers and fiber reinforcements, as well as predict post-moulding phenomena such as part warpage, also simulate material flow and cure of reactive moulding processes.[6]

Autodesk mould flow also offers the world's largest material database of its kind with more than 7,800 thermoplastic materials characterized for use in plastics CAE analysis, as well as thermoset materials, coolants and mould materials, and injection moulding machine-specific analysis capabilities.

Moldflow is an effective tool for comparing the implications of various gate designs, including:

- Gate location
- Molding window size at the gate location
- Filling pattern
- Gate size based on shear rate

3.1 Best Gate location

Gate is of great importance to part quality and productivity. It is defined as the small opening trough which the molten plastic material enters the cavity. Gate design for a perticular applications includes selection of gate type, dimensions, and location.

Best gate location is generated by gate location analysis. This is done by mold flow software by giving the input as meshing the component and material input parameters. Most suitable area is coloured blue which shows the best location for gate and which coloured red is the worst. A best gate location rating does not necessarily mean that the part can be filled from this location. This must be checked by running the analysis.[7,8]

This results help for the improving the efficiency of the mold design. The quality of the products and reducing the cost.

3.1.1 Edge (Standard) Gate

An edge gate is located on the parting line of the mold and typically fills the part from the side,top, or bottom. The edge or side gate is suitable for medium and thick sections and can be used on multi-cavity two plate tools.

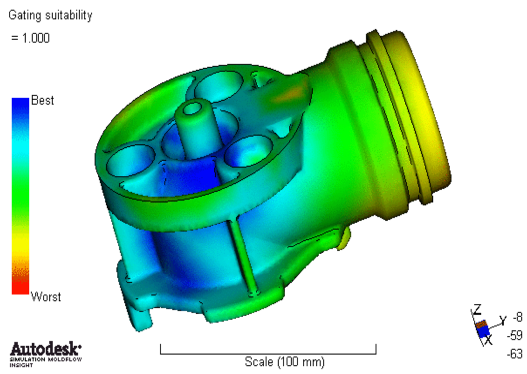


Fig 4: Best gate location for Manifold

Edge gate is used for component-manifold part, because it has thick sections.

3.1.2 Submarine (Tunnel, Chisel) Gate

A submarine (sub) gate is used in two-plate mold construction. An angled, tapered tunnel is machined from the end of the runner to the cavity, just below the parting line. As the parts and runners are ejected, the gate is sheared at the part.

If a large diameter pin is added to a non-functional area of the part, the submarine gate can be built into the pin, avoiding the need of a vertical surface for the gate. If the pin is on a surface that is hidden, it does not have to be removed.

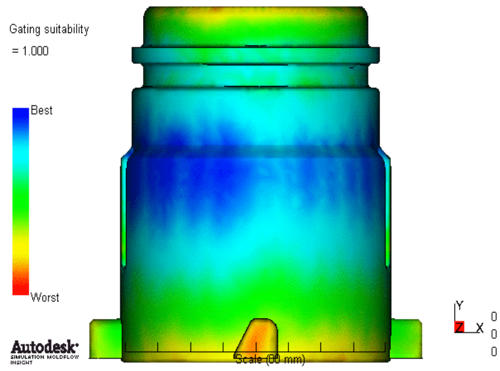


Fig 5: Best gate location for Coupler Body

Submarine gate is used for component- coupler body part, because it is thinner component and for easy degating.

3.2 Data Input

Input parameters are to be taken are the following for the analysis of the components.

3.2.1 Data Input For Manifold

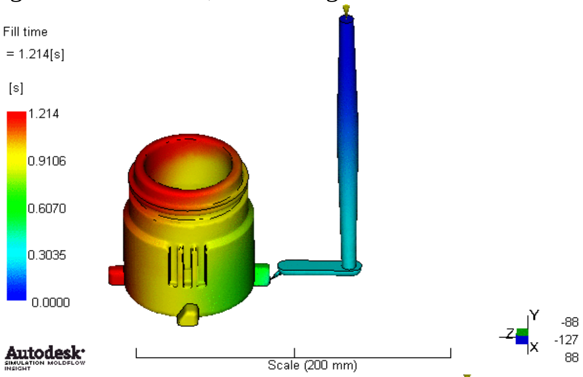
Material	POM-BASF-ULTRAFORM-H2320 006 UNC Q600
Mold temperature(°C)	80
Melt temperature(°C)	210
Types of analysis	Fill,flow,wrap
Types of mesh	3D
Feed system	Cold runner according to mold design
Cooling system	According to mold design

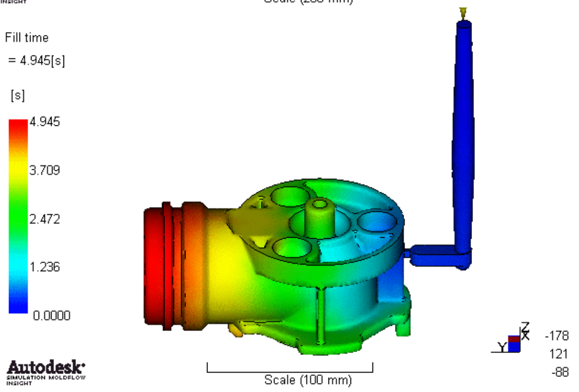
3.2.2 Data Input For Coupler Body

Material	PA6-GF15-GLS_POLYMERS-PG39-111
Mold temperature(°C)	70
Melt temperature(°C)	235
Types of analysis	Fill,flow,wrap
Types of mesh	Dual Domain
Feed system	Cold runner according to mold design
Cooling system	According to mold design

3.3 Fill Time

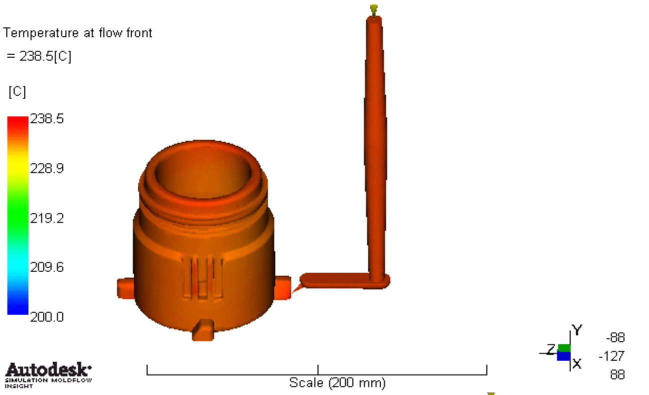
This result shows the flow path of the material through the part by plotting contours which join regions filling at the same time. These contours are displayed in a range of colors from red, to indicate the first region to fill, through to blue to indicate the last region to fill. A short shot is a part of the model that did not fill, and will be displayed as translucent. By plotting these contours in time sequence, the impression is given of material actually flowing into the mould. Multi Gate Bidirectional Filling -No deformation, Entire length utilized.

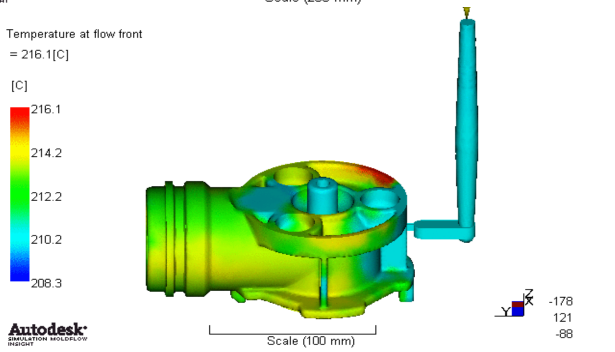




3.4 Temparature At Flow Front

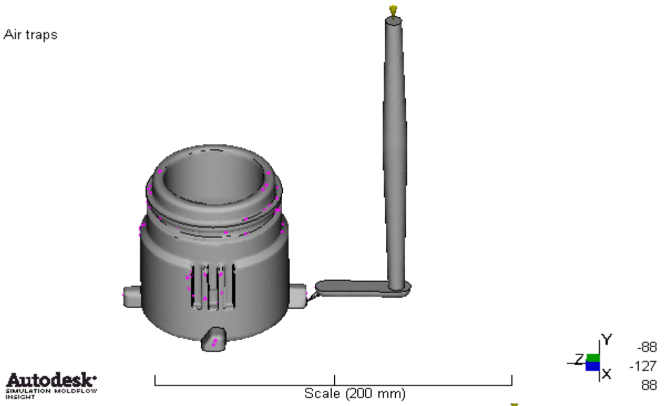
The flow front temperature result uses a range of colors to indicate the region of lowest temperature (colored blue) through to the region of highest temperature (colored red). The colors represent the material temperature at each point as that point was filled. The result shows the changes in the temperature of the flow front during filling.

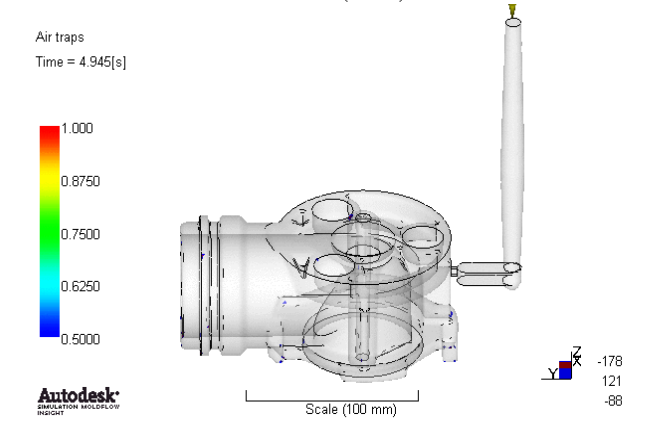




3.5 Air Traps

The air traps result is generated at the end of a Mid-plane or Fusion flow analysis, and shows a red line wherever an air trap is likely to occur. This may be where the melt stops at a convergence of at least 2 flow fronts, or at the ends of flow paths.



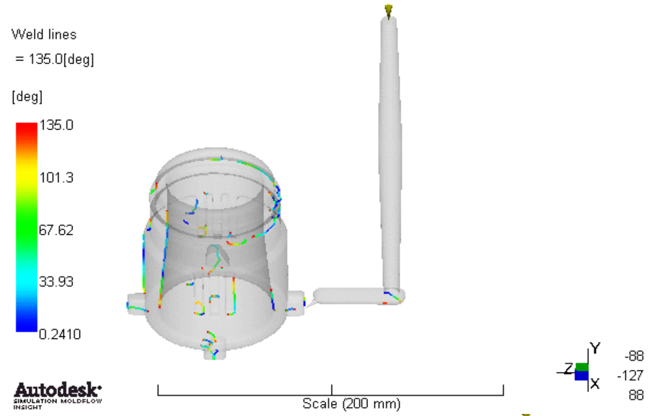


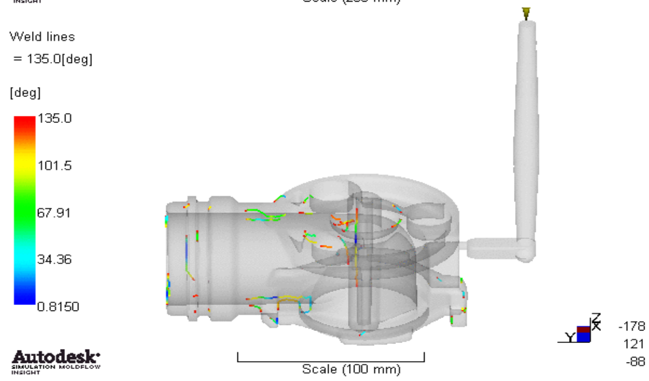
3.6 Weld Lines

When a weld line forms, the thin frozen layers at the front of each flow path meet, melt, and then freeze again with the

ISSN:2319-6890)(online),2347-5013(print)
01 July 2014

rest of the plastic. The orientation of the metal at the weld is therefore perpendicular to the flow path. This result indicates the presence and location of weld and meld lines in the filled part model. These are places where two flow fronts have converged. The presence of weld and meld lines may indicate a weakness or blemish. Increasing the injection pressure and adjust cooling water temperature in that zone is likely to avoid weld lines and give us good component. As this problem was occurred in the similar designs and those problems are overcame by slight adjustment in the process parameters. Multi Gate Bidirectional Filling - Cold weld-line formed at the Joint.





4. RESULTS AND DESCUSSION:

Regarding air traps we notice that we have optimum or the less air traps and these are to be removed by providing air vents. Time required to fill the mold are 4.9 sec and 1.214 sec for manifold and coupler body respectively. Temperature at the flow front are to be noticed are 216°C and 238°C for manifold and coupler body respectively, and lastly considering weld lines which are to be defective and forms the marks on the surface of the component, These are observed less.

Hence we select the data for processing and design of mold tool as per our calculations.

5. CONCLUSION

From the analysis we conclude that:

- Best location of the gate obtained from the analysis has been selected for manufacturing.
- The present gate size and location is finalized after balanced filling and other factors.
- The temperature at flow front in the most region of the component varies from 200 to 238°C which is close to the melt temperature of the polymer binder (234°C), so formed weld line has little effect on the strength of the product.
- Most of the air traps are located on the surface, which can be removed by providing suitable air vents in the mould.

6. References

i. Shen Changyu, Wang Lixia , Li Qian "Optimization Of Injection Molding Process Parameters Using CombinationOf Artificial Neural Network And Genetic Algorithm Method" Journal Of Materials Processing Technology 183 (2007) 412–418

ii. Vishnuvarthanan, M., Rajesh Panda, Ilangovan, S. And Nayak, S.K "Optimization Of Injection Molding Cycle Time Using Moldflow Analysis" International Journal Of Current Research Vol. 4, Issue, 01, Pp.259-261, January, 2012

iii. B. Duleba1, F. Greškovic2 "Application Of Ca Systems At Design And Simulation Of Plastic Molded Parts" Duleba Et Al., International Journal Of Advanced Engineering Technology

iv. Kamal, R.: Injection Molding Technology And Fundamentals. 1st Ed. Munchen, Germany: Hanser Publications, 2009, Isbn: 978-3-446-41685-7

v. Dulebová, L. :Vplyvmnožstvaregranulátunazmrštenievýstrekov ZPbt. In: Plasty A Kaucuk. Vol. 48, No. 9-10 (2011), P. 268-271. Issn 0322-7340

vi. M. Stanek, D. Manas, M. Manas And O. Suba "Optimization Of Injection Molding Process". International Journal Of Mathematics And Computers In Simulation.

vii. Plastics Insight, Xxxx. Moldflow Corporation, 91 Hartwell Avenue, Lexington, Ma, 02421.

viii. M. Vishnuvarthanan, Rajesh Panda And S. Ilangovan "Optimization Of Injection Molding Cycle Time Using Moldflow Analysis". Middle-East Journal Of Scientific Research.