ISSN No:-2456-2165

Flow Simulation Analysis for Inserted Thermoplastic Injection Moulding Part Connecting Plate

Thanusha Nansish*
MTech in Tool engineering
Government tool and training Centre.
Mysore-16, India

L.G.Sannamani
MTech in Tool engineering
Government tool and training Centre.
Mysore-16, India

Abstract:- Thermoplastics injection moulding is one the advanced method ,which is used to produce plastic components which are seen in daily use and also many of engineering components. Producing an plastic component is of the challenging task where lots of parameters should be considered with respect to its design, aesthetic and it mainly depends upon functional requirement & customer satisfaction. The prime cost consideration is given to the part quality requirement, which decides the method and process of manufacturing.

As per studies one of the main problem faced by the designer is flow of polymer from the injection location to fill complete mould with avoiding some of the defects like weld lines, air traps which indirectly affect the strength of the mould when load is applied. Here, from the use of mould flow software some of the parameters are studied which helps in selection of the machine type and also gives an clear idea to designer where the injection location should be selected for complete filing and also to avoid the defects. The analysis was carried out to known the gate location, fill time, weld lines air trap and pressure requires to complete fill of mould for the component know as connecting plate.

Keywords:- *Thermoplastic Injection Moulding, Mouldflow Analysis, Weld Lines, Fill Time,Air Trap.*

I. INTRODUCTION

Insert molding is a process in which plastic is injected into a mold that contains an pre-placed insert [1]. Inserts can be of metals or different types of plastic. The result of insert molding is a single molded plastic piece with an insert surrounded by the plastic. Many industries now a days use insert moulding. Plastic insert molding is ideal for improving the strength and reliability of a component, which also save on cost and production time by eliminating the need for secondary operations such as welding, polishing, soldering.

To produce good & high quality plastic injection molded parts, you need high quality plastic injection tooling and molds.

When it is said as injection mould there are parts in single mould which should be designed in perfect manner such as top plate, core and cavity inserts, ejection system feed system and some standard parts to align them all together.

When it comes to feed system the runner and gate used in plastic injection molding is responsible for directing molten plastic from the nozzle of the injection machine into the core and cavity insert of the tool.

The gate design and its location affects cycle time, tooling costs, lead time, the location of witness marks (weld lines, sink marks) and many other potential defects that are seen in injection moulding

Every designer who wants to develop a new product should know about the basics of gate designing. Knowing this information will help you to make informed decisions and also in selection of moulding machine.

- *Gate Location*

When gate location is to be considered there are some factors to be known, size of the gate, design of the gate and how many gates should be provided with respect to wall thickness of component. These are the main three factors which affects the process and the product.

The gate size is a measuring act between filling the mold exactly and keeping the smallest runner possible to improve shearing when the mold cycles in the machine[3]. While considering the gate appearance, by experience and outcome of many others plastic components, the nicer appearance of the product will be seen by providing the small gates, which also have benefit for easy degating.

Gate location should be given in such manner, so that defects like shrinkage and voids should get decrease.

A. Experimental analysis

Analysis was carried out with the help of mould flow software to get an clear picture of how component fills and what are defects should be taken care.

Selected part has many inserts with varying wall thickness which gradually leads to many problems during moulding.to avoid these problem the component is analyzed and results are seen for optimization in the process.

The software used is mould flow plastic insight, in which number of iteration was conducted to get best result and one of the iteration which gave suitable output has been discussed further.

This software utilizes CAD files and advance FEA analysis technique for better results in mould designing and for optimization of process parameters

B. Component And Material Study

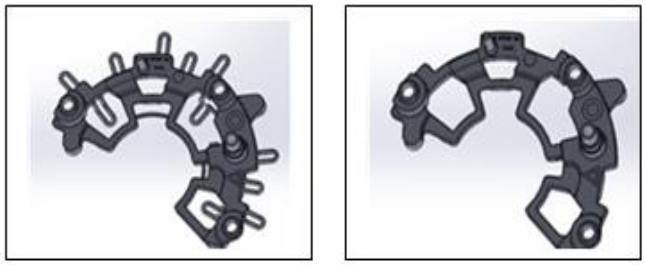

Fig 1:- 3D component with and without insert.

Component seen in above figure is known as connecting plate which is used in automobile parts where one can observe in fig[a]&[b] an inserts are placed in between the component whereas analysis is conducted without an insert.

Material selected for the component is an important factor as it gives life and functional aspects of the component with 33%of glass filled. Plastic material selected for this component is PA66. Which as a flame retardant property which prevents electric short circuits, strong and tough, and rigid.

"PA66"which is considered to be the most versatile polymer available with application both as plastic and fiber in virtually plastic end use market [7]. It exhibits high impact strength, high flow and are have high flame retardation.

C. Best gate location

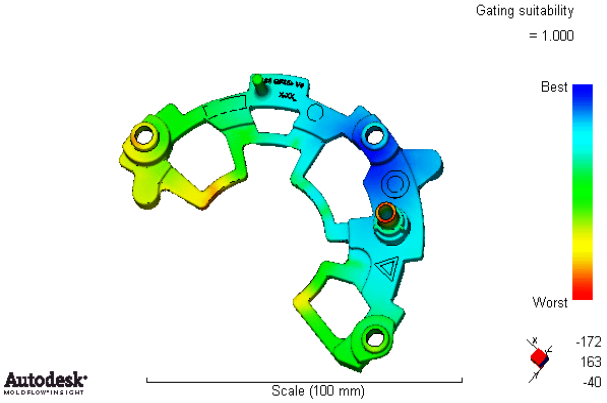

Fig 2:- Gate location

The best gate location is considered by the rated location after the analysis. The results are read by seeing the color code in which red indicates the worst place for gating and blue indicates best gating area, from the fig(2),one can observe the different color indication in between blue and red which tells that location of gate will be pretty good if it is chose in different location rather than positioning it near blue or red colored area .

The designer will consider the best gate location not only by the color indication but also knowing the other possibility when there is any inserts are to be placed inside the component or area in which machining should not be a problem. When the component study was done ,there were 4 inserts which are wire forms as to be placed in between ,so taking the other parameters in consideration like pressure, fill time etc., the best gate location was selected. The gate location should be given in such manner so that the flow of plasticizing polymer will be easy and uniform .to get the best result of the product, optimal location of gate and quality of the part should be considered, here 3 gating is given to the component for minimizing the size of the gate and also to lower the pressure ,which tends to affect the movement of the inserts placed inside.

D. Fill Time

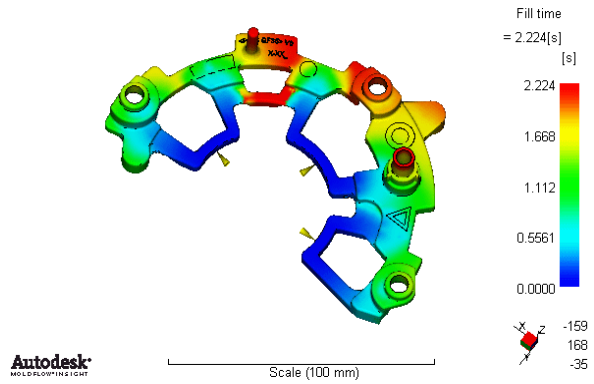

Fig 3:- Fill time

Fill time is the overall time taken by the polymer to completely fill the core and cavity.It is one of the important parameter to be considered, and also it indicates how quick the polymer is injected into the mould. Fill time affects how much shear thinning and heating the plastic experiences, which in turn affect the viscosity of the material , the pressure and temperature of the plastic inside the cavity, and the overall part quality. Variation in fill time may adversely affect the final product. Therefore once the ideal fill time is established for a given mold, that fill time should live with the mold forever and should be allowed to vary only slightly (±0.04 sec, as per John Bozzelli's recommendation) [2].

Here from the fig [3] the fill result is 2.224sec which is the time taken to fill the core and cavity completely therefore the short fill is not seen.

ISSN No:-2456-2165

E. Pressure at end fill

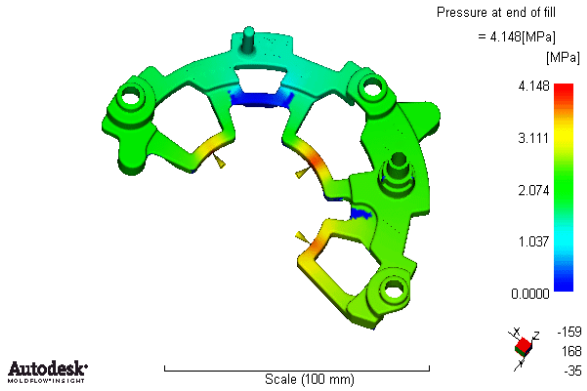

Fig 4:- Pressure at end fill

From fig [4], the result obtained gives the maximum pressure required to completely fill the core and cavity, which is 4.418 MPa, by which the selection of injection moulding machine can be decided.

Pressure at end of the fill gives an actual idea on pressure drop on each section of the part from the injection point, the variation of the pressure on each surface with flow of polymer gives the confident of mould fill. From where the injection location is given it is observed how the drop in pressure happed, the color variation is seen and observed where red color represents high pressure drop and blue indicates the lowest drop in pressure.

If the fill pressure is high or low, then defects are seen such as mould flashing, and affect the surface finish of the component. Hence for the better surface finish fill pressure should not be close to the given machine pressure.

F. Weld Lines

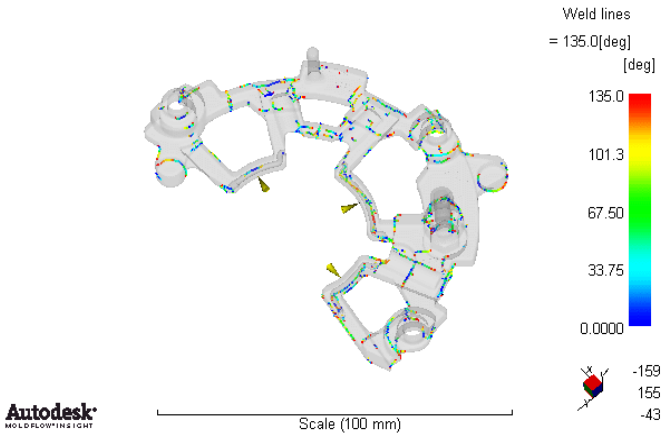

Fig 5:- weld lines

Weld lines are one among the defects which is studied in injection moulding process where it cannot be eliminated completely but can be reduced by taking certain design consideration which should be taken care by the designer.

Weld lines inversely effects on the strength of the part and also the appearance. One of the most difficult defects to eliminate.

The origin of the weld lines is also by the choice of the material, design and its structure of the part. They occur where polymer flows meet/collide at the same point from different flow direction. Thus resulting in the formation of weld lines.

The component study was done in detail and the surface on which more force acts was also noted, by considering that it is observed in fig [5] how the weld lines are formed in different region. Where there are weak weld lines and also the strong weld lines, these can be decreased by changing the injection location or redesign of gating.

G. Air Trap

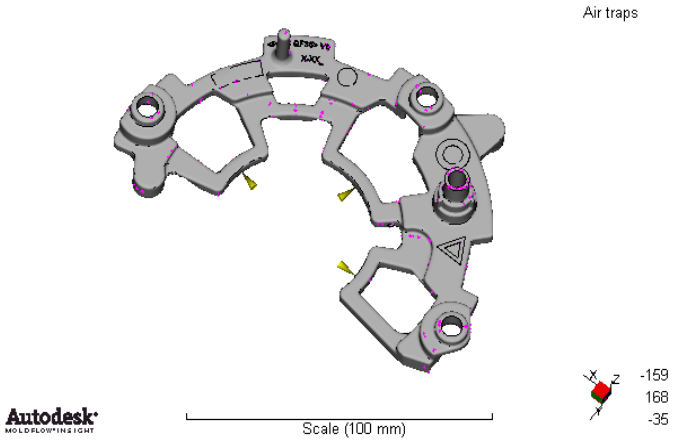

Fig 6:- Air traps

From the fig [6] the pink region or dots that are observed are the regions where air has been trapped this occurs because air fails to escape from the vents or inserts provided in mould.

Air traps occurs in the region where the flow of polymer tends in preferentially thicker regions in component or region where the sections are critical in design, which results in voids and bubbles inside the component and then makes final product with incomplete filling.

When analysis was observed from the mould flow software the results were studied and decided to provide suitable air vents, so that air trapped are forced to escape from the mould it can also be reduced by changing injection speed ,properly placing the vents in mould or enlarging the previously given vents.

II. DISCUSSION AND CONCLUSION

From the study ,and observation of an inserted type of component some of the results were taken by the help of mould flow analysis software ,in which selection of the best gate location gives an detail view of many results like fill time, temperature to fill, shear stress, volumetric shrinkage, air traps weld lines etc. from which some results are studied in detail

- Best gate location gives an idea where the injection location can be given in a component.
- Fill time obtained was 2.224sec which helps in setting the machine parameters.
- Pressure at end fill gives a detail view where pressure drops from the injection point to the end of the fill can be seen, where maximum pressure for complete fill can be noted.
- Due to the temperature of the mould weld lines formed will get fused resulting in good surface finish.
- Air traps can be eliminated by giving suitable vents in the mould.

Analysis was conducted on connecting plate using mould flow software, for investigating the process parameters. Filling time required and suitable gate location was know which helps designer to take decision for providing perfect injection location and also the defects which was observed helps in varying the mould temperature and what are the measures to be taken in mould to avoid these defects, all this study results in high quality product.

REFERENCES

[1]. https://www.tuthillplastics.com/services/insert-molding.
[2]. https://www.ptonline.com/articles/a-new-look-at-evaluating-fill-times-for-injection-molding
[3]. https://www.midstatemold.com/gate-design-and-gate-size-in-injection-molding
[4]. G. Menges, P. Mohren, How to Make Injection Molds, second ed., Hanser Publishers, New York, 1993, p 129.
[5]. R.J. Crawford, Rubber and Plastic Engineering Design and Application, Applied Publisher Ltd., 1987, p. 110.
[6]. *M. Vishnuvarthanan, Rajesh Panda and S. Ilangovan* optimization of Injection Molding Cycle Time Using Moldflow Analysis .
[7]. http://www.aquentpolymers.com/pp-glass-field-granules.html