# Effect of Feeding System on Thin Walled Injection Molding

**Kartik T* and Rajesh R**
Department of Production Engineering, PSG College of Technology, Coimbatore – 641 004
***Corresponding author: E-Mail: kartikt008@gmail.com**

**ABSTRACT**

Injection molding is a near net shape manufacturing technique for the production of plastic components. The paper takes up the problem encountered in the manufacturing of thin walled components. Thin walled components have a wall thickness in the region of 200 – 300micrometers. The major defect in the injection molding of thin walled component is incomplete filling of the mold cavity especially in the thin walled section. Simulation of injection molding is carried out using Mold flow software to identify the defect and to rectify it. With the aid of computer aided engineering (CAE) software conventional trial and error method of optimizing is avoided. For reduction of defect feeding system to the cavity is altered.

**KEY WORDS:** Injection molding, Moldflow, feeding system.

## 1. INTRODUCTION

Plastics are the most widely used material, though are relatively new when compared to other materials, surpassing world's consumption of steel, aluminum, rubber, copper, and zinc. Injection molding is a repetitive process in which melted polymer is injected into a mold cavity, packed under pressure, and cooled until it has solidified enough. As a result, it duplicates the cavity of the mold. The mold consists of a single cavity or a series of similar or dissimilar cavities, connected with each other through runners that direct flow of melt to the individual cavities.

The injection molding process is of great significance as it can produce finished, multifunctional, or complex molded parts accurately and repeatedly in a single, highly automated operation. It permits mass manufacture of a great variety of shapes, from simple to intricate three-dimensional products and extremely small to large. When required, these products can be molded to extremely tight tolerances, very thin, and in weights down to milligrams. Figure 1 illustrates mold cavity details consisting of four disc type products being connected by feeding system to be injection molded in a single cycle.

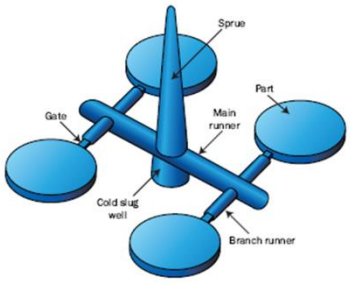



**Figure.1. General arrangement of mold**

**Literature Review:** Plastic products are commonly used for their ease to manufacture, light weight and low cost. A lot of research is carried out in this field to overcome the problems faced in the processing of the plastic products. Injection molding is the manufacturing technique used for producing plastic components. Thin walled components pose an additional challenge which is a current field of research for many. Yung kangshen (2008). have worked on the gating system optimization for the thin walled components. Through simulation work he has confirmed that thin walled components can be manufacture by adjusting process parameters and by providing a suitable gating system. Muralidharan (2016), have developed mathematical model for the sprue design using computer.

Oroszlány (2010), found the influence of gate type on thermal characteristics of injection molded components. The work undertaken by them demonstrates change in the filling and deflection characteristics of injection molded screws with the use of symmetrical gate and ring gate. Lee Tin Sin et al. have used computer-aided injection molding process analysis for optimal mold design. They have conducted a comparative study of different material to suite product. Main objective of the experimentation was to reduce production time and shrinkage of the component.

Sanchez (2008), have designed the sprue for achieving quality PVC products. For this they have changed feeding system to analyze process through the design of two different sprues. They have conducted experimentation with 1° and 3°sprues to overcome blush defect in PVC products. Through simulation and experimentation they have found 1°sprue has fewer defects due to its rounded end that makes the flow enter into the mould in a softer way, and the white mark disappears. Rounding the end of sprue to get a better surface appearance of the part.

**Problem Definition and Objective:** Thin wall section injection molding is a complicated process. Thin walled components have a wall thickness in the region of 200 – 300 micrometers. Due to very small cross sectional area for

the flow of melt number of defects arises in their production. Hesitation is a defect that results from the stagnation of polymer melt flow over a thin-sectioned area or area of abrupt thickness variation.

The work under taken is specific to the hesitation defect encountered in thin walled sections. The objective of the study is to find the optimal process parameters for injection molding of thin walled component. With the use of CAE software – Moldflow which is injection molding simulation software predict process variables to eliminate defects.

**Simulation and Analysis:** Plastic injection molding simulations are carried out using Mold flow software. Mold flow analysis is carried in initial design stage, with mold designed for the optimum filling pattern to avoid defects in the actual process. It is Finite Element Application (FEA) based software which divides the object into fine elements and applies boundary condition to obtain end result. Component is imported in the IGES format and meshed to divide the component into elements with nodes. Based on values of the elements component details can be obtained by extrapolating.

**Problem simulation trial without feeding system:** The component is modeled in Pro-engineer and converted to Initial Graphic Exchange Specification (IGES) format for compatibility with Mold flow software. The component under study is battery cover with outer wall thickness of 1mm and inner wall thickness of 0.3mm. This is an ideal component for the study of thin walled injection molding process. Initial simulation trial is conducted for component without feeding system to analyze defect and for gate location details.

Component is imported in the IGES format and meshed to divide the component into elements with nodes. Based on values of the elements component details can be obtained by extrapolating. Material and process details for the simulation are listed in the table.1 and 2.

**Table.1.Material detail.**

| | |
|---|---|
| **Commercial name** | VB1108R |
| **Manufacturer** | Cheil Industries Incorporated |
| **Family abbreviation** | PC+ABS |

**Table.2.Injection process details**

| | |
|---|---|
| **Mold temperature** | 75°C |
| **Melt temperature** | 265°C |
| **Melt temperature** | 230 -300 °C |
| **Ejection temperature** | 117 °C |

Injection point for the meshed component is taken at outer boundary as it has more thickness. Fill analysis is done without feeding system and result shows a defect called hesitation is formed in the product (figure 2).

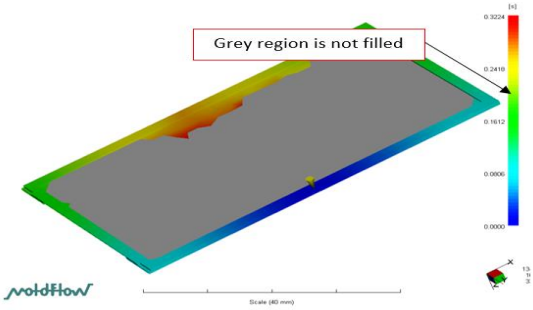



**Figure.2.Central region is affected by hesitation**

**Simulation trials with feeding system:** The simulation is carried out with different number of gates and location to find the optimal filling condition. The gate locations are determined based on trial run with gating system. The trials are conducted with1, 1+1 (one on either side), 2, 2+2 (two on either side), 3 and 3+3 (three on either side) gating configurations.

## 3. RESULTS AND DISCUSSION

Fill analysis is done without feeding system and result shows a defect called hesitation is formed in the product. This defect is common in thin walled components.

**Hesitation:** Hesitation is a defect that results from the stagnation of polymer melt flow over a thin-sectioned area, or area of sudden thickness variation. Hesitation can be eliminated by changing part thickness or shifting the gate location.

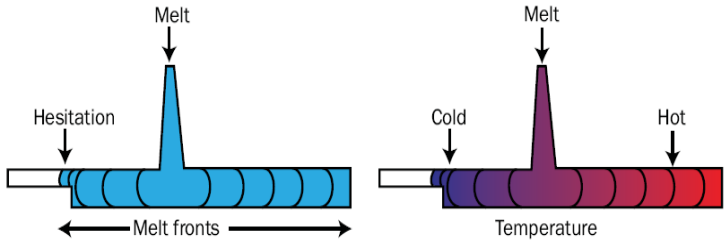



**Figure.3. Schematic representation of hesitation defect**

When melt is injected into a cavity of variable thickness, it tends to fill the thick and less resistant areas. As a result, melt may stagnate at thin sections until the rest of the part is filled and the stagnated polymer melt starts moving again (figure 3). If the duration of hesitation is significant, polymer will solidify prematurely at stagnated point. When solidified melt front is pushed to the part surface, a surface defect such as a hesitation mark occurs.

To avoid hesitation defect the mold is pre-heated to 200°C. The simulations justify the same. The following graph (figure 4) is plotted between pre-heated temperature of the mold and mold fill percentage without gating system.

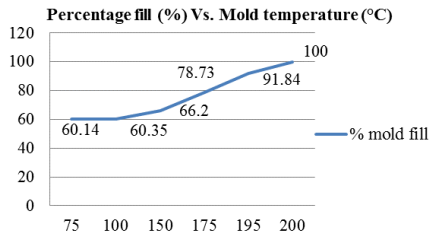
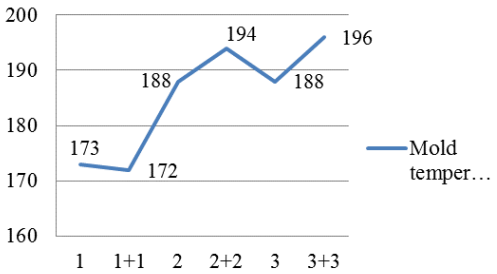



**Figure.4. Percentage fill vs. Mold temperature graph without gating system**

**Feeding system simulation trials:** In simulation trial without feeding system, it is found that for the component to be filled mold temperature has to be raised to 200°C. By raising temperature, viscosity of the molten plastic is lowered aiding in easier flow. But the temperature is too high can lead to the degradation of plastic.

To lower mold temperature, feeding system is designed for directional filling of mold. Fill time and mold pre-heating temperature parameters are plotted against number of gating locations in figure 5 and 6. Some of the observations made in the graph are as follows:

- Gate 1+1 configuration has the least mold pre-heat temperature and above it the temperature increases drastically.
- Fill time graph clearly shows the 1 gate configuration is best with the least fill time.
- Based on the two graphs 1 and 1+1 configuration gates are more apt for the component under study. But 1+1 configuration is selected for better melt flow pattern.



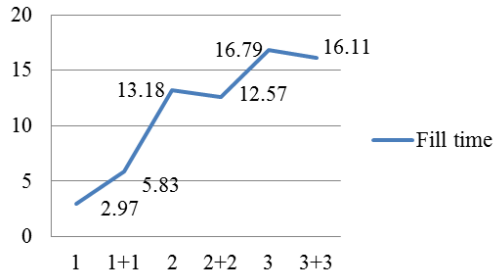
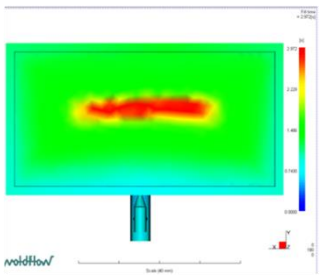
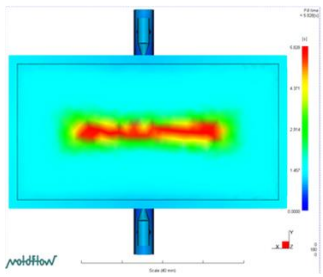




**Figure.5. Mold temperature (°C) Vs Gate number** **Figure.6. Fill time (s) Vs Gate numbre**

Fill time simulation results are shown in figure 7. The region in red are filled at last. For all the configurations results are similar with the outer thicker region being filled first and then melt flow into the central region which is filled at last.



**Configuration 1**



**Configuration 1+1**

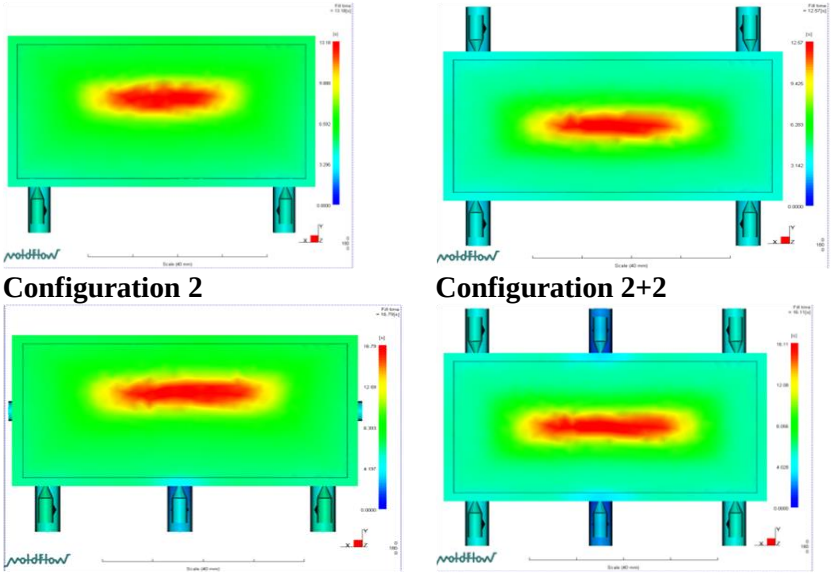



**Figure.7. Fill time results**

Based on the comparative study of cold and hot runner system graphs (figure 8 and 9) are plotted for fill time and mold pre-heating temperature. Some of the observations made in the graph are as follows:

- Hot runner system has better fill time and temperature characteristics. So based on the cost requirement for production the correct system can be selected.
- Hot runner system with 1+1 configuration is showing better results compared to the cold runner system which can be taken as the optimal result.
- Cold runner with configuration 1+1 has same mold pre-heat temperature as that of hot runner system but its fill time is higher.

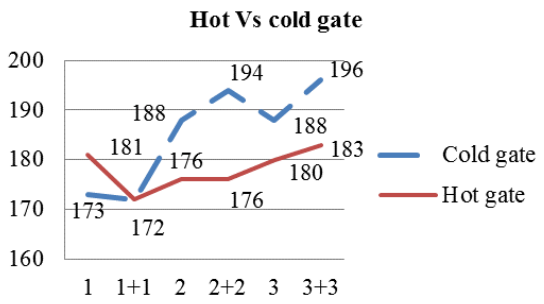



**Figure.8.Cold gate Vs hot gate mold temperature (°C) comparison graph**

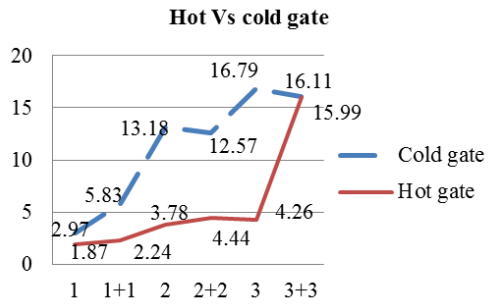



**Figure.9.Cold gate Vs hot gate fill time (seconds) comparison graph**

## 4. CONCLUSION

The simulation results clearly indicate that cold runner with the feeding configuration of 1+1 has the best result. Hot runner system does give the added advantage of lower fill time but the cost implication is high. So the final decision can be made based on the overall cost of the component.

The role of CAE software in the design of molding process has to be appreciated for reducing cost and time which would have been wasted if conventional trial and error type experimentation had been taken up.

## REFERENCES

Alfredo Campo, The complete part design handbook – for injection molding of thermoplastics, Hanser Publishers, 2006.

Huamin Zhou, Computer modeling for injection molding, John Wiley & Sons publication, 2013.

Jay Shoemaker, Moldflow Design Guide – A Resource for Plastics Engineers, Moldflow Corporation, 2006.

Lee Tin Sin, Rahman W.A, Rahmat A.R, Tiam-Ting Tee, Soo TueenBeea and Low Chong-Yu, Computer aided injection molding process analysis of polyvinyl alcohol–starch green biodegradable polymer compound, Journal of Manufacturing Processes, 14, 2012, 8-19.

Muralidharan Lakkanna, Mohan Kumar and Ravikiran Kadoli, Computational design of mold sprue for injection molding thermoplastics, Journal of Computation Design and Engineering, 3, 2016, 37-52.

Oroszlány and Kovács J.G, Gate type influence on thermal characteristics of injection molded biodegradable interference screws for ACL reconstruction, International Communications in Heat and Mass Transfer, 37, 2010, 766-769.

Sanchez B and Llado J, Surface quality of PVC fitting based on the design of the sprue, Journal of material processing technology, 207, 2008, 13-20.

Yung-Kang Shen, Chih-Wei Wu, Ya-Feng Yu and Hung-Wei Chung, Analysis for optimal gate design of thin-walled injection molding, International Communications in Heat and Mass Transfer, 35, 2008, 728-734.