International Journal of Applied Engineering Research
ISSN 0973-4562 Volume 10, Number 11 (2015) pp. 27817-27826

http://www.ripublication.com

Study on Parameters Influencing Fill Time In A Multi-Cavity Mold

Anoop.K.Philip[1], M.Elangovan[2]
[1]M.Tech. Student, Department of Mechanical Engineering, Amrita School of Engineering
[2]Professor, Department of Mechanical Engineering, Amrita School of Engineering
Amrita Vishwa Vidyapeetham, Coimbatore –641112, Tamilnadu – India
[2]m_elangovan@cb.amrita.edu

Abstract

Plastic injection mold design is a highly iterative process. Each and every iteration is highly time consuming and costs considerably for the mold maker. CAE tools help mold designers in their iterative design process and to reduce the cost and time. Optimization of fill time in injection moulding has a very important role in improving the productivity but without sacrificing on the quality of the product. Analysis and study of a multi-cavity mold is more challenging as it involves balancing of multiple requirements. The effect of process parameters on fill time along with the design of runner system for the multi-cavity mold has been focussed in this paper. A representative product, like a coat button as a component has been studied and analysed with the help of Autodesk Simulation Moldflow Adviser 2014. The results of the analysis has been discussed and reported.

Keywords: Injection molded component, Fill time, Mold flow analysis, Parameter optimization

Introduction

Plastics has been considered one of the most preferred materials among designers. Conventional materials such as wood, metals, stones, leather etc. are replaced by plastics, due to their unique properties such as light weight, corrosion resistance, recyclability, chemical resistance, low cost, ease of manufacturing, versatility etc.[1]. Plastics can be manufactured by different processes which include extrusion,blow moulding, fabrication, casting, compression moulding,foaming, injection moulding, rotational moulding, thermoforming of sheetetc.[2].

Injection moulding is one of the widely used manufacturing technique in processing of plastics because of its advantages such as ease of manufacturing, fast production, material and colour flexibility, less cost of production, flexibility in design and low waste [3].It consists of mold filling, packing, and cooling stages [4].Complexity in design of part and mold has turned injection molding to a highly iterative and time consuming process. As a solution to the above complexity CAE tools are developed with variety of applications which can increase the easiness in designing of part and mold. CAE tools has been transformed as an inevitable part of today's world due to its economical nature and reduced time consumption [5]. Requirement of plastic parts with good quality are still on the increase in automotive sector[6].Quality not only depends on the plastic material but also on process parameters. So manufacturing process conditions must be optimised in order to get a good quality [7, 8].

Product cost mainly depend on design, material, and processing expenses [9]. Processing cost can be reduced by reducing the cycle time and increasing the number of parts produced per cycle. Molding conditions depend on important parameters such as melt temperature, fill time, packing time, and packing pressure[7]. Fill time is one of the factor which control cycle time, so reduction in fill time will also result in reduced cycle time. Runner design have an important role in the fill time variations. Many researchers indicated that runners play a significant role than any other factor in mold design [10-12].

This paper discuss about the various parameters which affect the fill time of a multi-cavity mold and the possibilities of reducing the fill time. Autodesk Simulation Moldflow Adviser 2014 was employed here to analyse and study about different factors.

Parameter Selection

There are various parameters that affect the fill time of a multi-cavity mould, such as runner length, runner cross section, part volume, gate location, runner diameter, injection pressure and speed, mould temperature, injecting temperature, sprue shape, gate shape, material used, runner type (hot or cold), cooling time etc. Six important parameters are selected here to study the effect of fill time and are described below:

A. Runner Diameter

Runner is the portion of the feed system that connects the sprue to the gate. It's an open channel machined in the Parting Surface. Designing of runner is very important in mold design. Runners make the way for the molten material to the cavity[13].

The designer must consider the following factors before deciding the size of the runner[13].

1. The wall thickness and volume of molds
2. The length of runner
3. Should concern about runner cooling
4. Availability of different cutters for machining the runner
5. Plastic material used

B. Runner Cross-section

The measure of effective runner design is that runner should offer maximum cross sectional area for efficiency of flow at the same time its perimeter is minimum, else the fill time will increase and hence cycle time also. The selection of runner cross section also depend on the possibility of positive ejection of the runner system. Positive ejection is not practicable for multi plate molds. Here the preferred shape of runner is basic trapezoidal type [13]. Fully round runner or hexagonal runner is preferred for simple two plate molds, which have a flat parting surface. This type of runner will increase the mold cost, because it requires machining of semi-circular runner on both half of mold plate. Compared to multi plate molds the increased mold cost is less in two plate system. It is difficult to match accurately the semi-circular channels of the round runner in case of complex parting surfaces, so trapezoidal or modified trapezoidal can use here[13].

Circular, circular tapered, rectangular, semi-circular, trapezoidal runner shapes are analysed in this work to find out the minimum fill time.

C. Runner Type

Injection molds can be classify based on the type of runner system using. They are mold with hot runners and cold runners[14].

i. Cold Runner

Usually this type of molds consists of 2 or 3 plates and that are located within the base of mold. When molten material is injected to the mold, the sprue will fill first and then runners which leads to the mold cavity[15]. Runner system and parts are attached in two plate molds and an ejector pin helps to detach the parts and runner from mold. In case of three plate mold runner located on separate plate and so the ejector pin ejects the parts alone[16]. To reduce the waste, runner is recycled in both system but this will increase cycle time.

ii. Hot Runner

In hot runner system, the runner is situated in a heated manifold which keep the temperature of runner above the melting point of the plastic material. Heated nozzles in the hot runner system directs the molten material to the cavity[15].

Hot runner will help to reduce or eliminate scrap and to reduce cycle time of injection molding process. The main disadvantage of hot runner is that it makes the mold more expensive [17].

D. Material

There are a number of injection molding resins available in the world market today. Selecting the correct material for a specific application is difficult and it requires in depth knowledge about the resins available[18]. Changing the material after mold design and construction causes variations in geometry, because different materials have different shrinkage rate[19]. Injection mold fill time also different for different materials. CAE tools will help us to decide the correct material with less fill time.

E. Gate location

The gate position should be in such a way that it never supports formation of weld lines, since weld lines formation weaken the part mechanically. It is possible to avoid weld line on parts having circular cross section by setting the gate position on the base or apex of the mold. Central gating is also suitable in cases where side gating causes deflection of core[13].

Edge/side gate is economical for most of the moldings. In case of rectangular molding side gate with overlap feeding is better to reduce the flow line formation [13].

F. Runner length

When the melt flows through the length of runner,thepressure loss as well as heat losses will make the filling of cavity difficult. So the length of runner should be as minimum as possible. Runner length to each mold cavity from the starting of sprue should be same. This is known as balanced runner system. It will help to reduce the fill time [13].

Component Details

The component selected to study about fill time is a coat button. Because buttons are a simple but inevitable part of clothing's, bags, and wallets that works like a fastener and comes in a variety of size and shapes, it was chosen to be a representative component for the study. The diameter of coat button that was chosen for the study in the range of 20-28mm [20].Table 1 shows a coat button and its dimensions.

Table 1: Component and Its Dimensions

Diameter		
	Outer	25mm
	Inner	19mm
	Inner hole	2mm
Thickness		
	Outer	3mm
	Inner	2mm

Modelling In Moldflow

The software used to analyse and optimize the fill time is a CAE tool named Autodesk Moldflow Adviser. This tool helps us to simulate filling and packing processes of injection molding and also to guess the flow behaviour of molten plastic. By this way it helps to achieve good quality products. Mold designers can evaluate processing conditions, balance runner systems, optimize gate locations and predict and correct molded part defects[21].

Since the component selected here is small in size, a multi cavity mold with eight cavities are considered so as to increase the production rate and to reduce cost of production. Figure 1shows the model which done in MOLDFLOW software and Table 2 describes the specifications of the model.

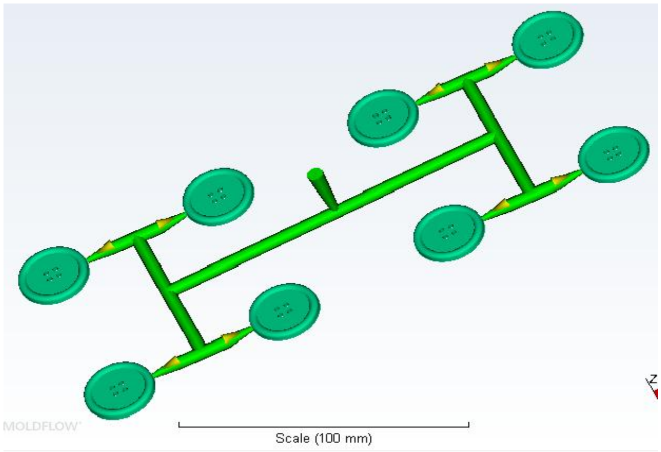

Figure 1: Eight cavity mould designed in MOLDFLOW

Table 2: Specifications of the model

Sprue		
	Type	circular tapered
	Start diameter	8mm
	End diameter	6mm
	Length	25mm
Gate		
	Type	circular tapered
	Start diameter	3mm
	End diameter	1mm
	Length	5mm
Number of cavity		8
Material		generic PP
Runner configuration		balanced
Volume of buttons		1030.83x8mm^3
Raw and column spacing		25mm
Gate location		side
Runner diameter		5mm
Runner cross section		circular

Results and Discussion

Simulation was carried out for various combinations to study the effects of runner and its allied parameters. The results are given below along with their description.

A. Runner Diameter

To optimize the runner diameter, it is varied from 1mm to 7mm. Table 3 shows the result obtained from analysis.

Table3: Relation Between Fill Time and Runner Diameter

Runner diameter	Mass of runner material(gm.)	Mass % of runner material	Quality of product	Fill time (s)
1	0.182	2.47	Bad	0.2375
2	**0.728**	**9.9**	**Good**	**0.2443**
3	1.63	22.17	Good	0.2613
4	2.913	39.58	Good	0.2949
5	4.55	61.88	Good	0.3396
6	6.55	89.09	Good	0.39
7	8.92	121.32	Good	0.4645

Table 3 shows that fill time is proportional to diameter. The runner diameter should be small to get a lesser fill time. But at 1mm diameter the quality of product was found to be bad. So the optimized runner diameter was chosen to be 2mm.

B. Runner Cross-section

To find out the best runner cross section with less fill time, 5 different runner cross sections selected and kept cross section area of all as same.

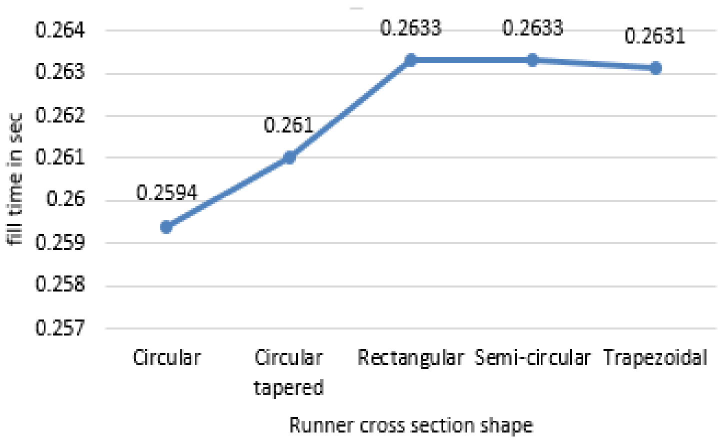

Figure 2: Runner Cross Section Shape And Fill Time Relation

From fig.2 it is clear that circular cross section has the least fill time than others.

C. Runner Type

To find out suitable runner type analysis was carried out with hot and cold runners. The analysis result shows that fill time is less for hot runners. This is because the metal flows at a higher rate when the runner is hot than when it is cold.

Table 4: Relation Between Fill Time and Runner Type

Runner type	Fill time
Hot	0.2161
Cold	0.2594

D. Material

To find out the material with lower fill time, 6 different materials were chosen that are commonly used in polymer industry. The analysis results of fill time with respect to material shown in the table 5.

Table 5: Relation Between Fill Time and Material

Manufacturer	Trade name	Family abbreviation	Fill time
Generic default	Generic PP	PP	0.2161
Husky	Mixed resin	PE	0.8123
Geon	80520	PVC	1.027
Idemitsu kosan Ltd	Styrol NF20	PS	0.2094
INEOS acrylics	LUCITE 40	PMMA	0.4245
Mitsubishi chemical	Yukon JX10-7	HDPE	0.3273
Asia poly	M201	LDPE	0.2158

From Table 5 it is clear that fill time is minimum for PS. Selection of one material depends factors such as price, availability, etc.

E. Gate location.

The fill time with respect to gate location namely, side gate and top gate, is analysed and the results are shown in table 6.

Table 6: Relation between fill time and gate location

Gate location	Fill time
Side	0.2161
Top	0.2151

From table 6 it is clear that optimum fill time is better at the top gate position.

F. Runner length

Runner length is the distance from end of sprue to the start of gate. In MOLDFLOW it was not possible to vary the length as it is, so the distance between the components was varied to vary the runner length.

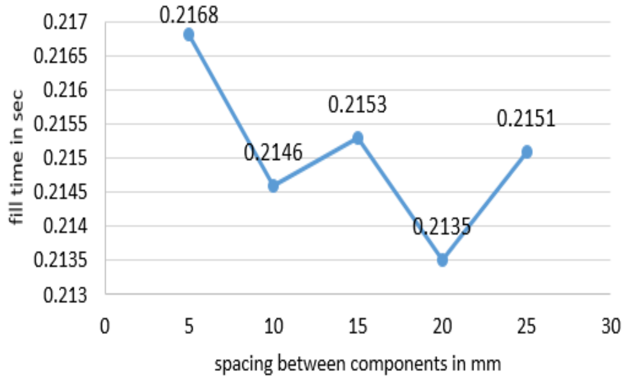

Figure 3: Relation Between Component Spacing And Fill Time

Fig 3 shows the relation between fill time and Spacing between components. The graph says that fill time is minimum at 20mm spacing between components

G. Optimized Parameters

The objective of this work is to find out a best possible runner design to achieve a minimum filling time. For that some parameters are selected and analysed to study the relation of the parameter and fill time. The best results and simulation model which got from this study are given below.

Table 7: Optimized Parameters

The parameter which varied	The best result is at
Runner diameter	2mm
Runner cross section shape	Circular
Runner type	Hot
Suitable material	PS
Gate location	Top
Spacing between components	20mm

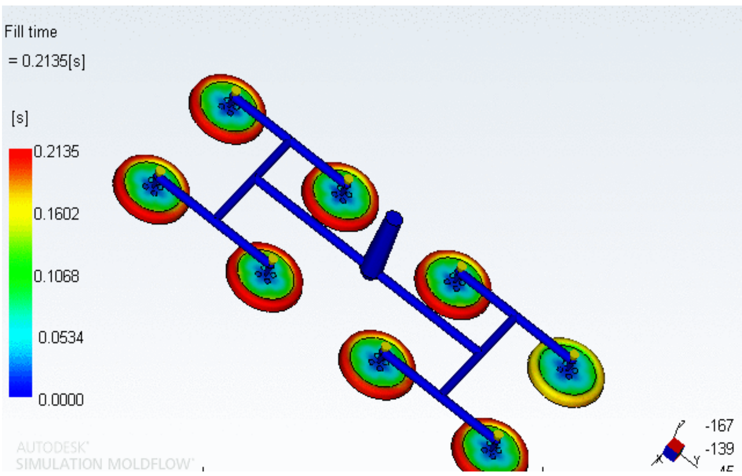

Figure 4: The Model With Minimum Filling Time

Conclusions

In the present era, traditional materials are being replaced by plastic components in all fields. With the help of CAD/CAE tools, it is now possible to develop a perfect mold for plastic industries with less iterations and less time.

In this work the main focus was to study the fill time of multi-cavity mold and the effect of various parameters that affect the flow. The model was designed and analysed in moldflow by importing it to the moldflow. Parameters such as, runner diameter, runner cross section shape, spacing between components, runner type were studied along with selection of suitable material, and gate location to get the best results.

The parameters in combination with the selected materials yielded in lower fill time of the representative component in a multi-cavity mold. The results of the analysis are reported for manufacture of the mold.

Further study can be carried out by varying the sprue dimensions, dimension and shape of gate, etc.

References

[1] (2014). *Plastic*. Available: http://en.wikipedia.org/wiki/Plastic

[2] (2014). *A - Z of plastics manufacturing processes*. Available: http://www.modip.ac.uk/resources/curators_guide/manufacturing_plastics

[3] M. Zhai, Y. C. Lam, and C. K. Au, "Runner sizing and weld line positioning for plastics injection molding with multiple gates," *Engineering with Computers,* vol. Vol. 21, pp. 218-224, May, 2006 2006.

[4] S. Jiang, Z. Wang, G. Zhou, and W. Yang, "An implicit control-volume finite element method and its time step strategies for injection molding simulation," *Computers & Chemical Engineering,* vol. 31, pp. 1407-1418, 2007.

[5] S. Chianrabutra, A. Wongsto, and T. Sirithanapipat, "Balancing gate and runner systems for a family mold using CAE tools," 2004.

[6] E. Bociąga, T. Jaruga, and J. Koszkul, "Plastic flow investigation in multicavity injection mold," *Proceedings of the 12th Scientific International Conference „Achievements in Mechanical and Materials Engineering" AMME,* pp. 107-110, 2003.

[7] X.-P. Dang, "General frameworks for optimization of plastic injection molding process parameters," *Simulation Modelling Practice and Theory,* vol. 41, pp. 15-27, 2014.

[8] B. Ozcelik and T. Erzurumlu, "Comparison of the warpage optimization in the plastic injection molding using ANOVA, neural network model and genetic algorithm," *Journal of Materials Processing Technology,* vol. 171, pp. 437-445, 2006.

[9] R. K. Irani, S. Kodiyalam, and D. O. Kazmer, "Runner system balancing for injection molds using approximation concepts and numerical

optimization," in *Proc 18th Annual ASME Des Autom Conf, Montreal, September*, 1992, pp. 255-261.

[10] S. Sulaiman and T. C. Keen, "Flow analysis along the runner and gating system of a casting process," *Journal of Materials Processing Technology,* vol. Vol. 63, pp. 690-695, Jan, 1997. 1997.

[11] J. C. Lin and C. C. Tai, "The Runner Optimization Design of a Die-Casting Die and the Part Produced," *International Journal of Advance Manufacturing Technology,* vol. Vol. 14, pp. 133-145, Feb, 1998. 1998.

[12] B. H. Hu, K. K. Tong, X. P. Niu, and I. Pinwill, "Design and optimisation of runner and gating systems for the die casting of thin-walled magnesium telecommunication parts through numerical simulation," *Journal of Materials Processing Technology,* vol. 105, pp. 128-133, Sep, 2000 2000.

[13] R.G.W.PYE, "injection mould design-an introduction and design manual for the thermoplastic industry," fourth ed london, 1989, pp. 135-167.

[14] (2014). *Hot or Cold Runner?* Available: http://intrepidmolding.com/news/hot-cold-runner/

[15] (2013). *Plastic Injection Molding 101 - Cold runner versus hot runner molds*. Available: http://info.rodongroup.com/blog/bid/95707/Plastic-Injection-Molding-101-Cold-runner-versus-hot-runner-molds

[16] (2014). *Cold Runner Vs. Hot Runner Molding Systems*. Available: http://nanomoldcoating.com/cold-runner-vs-hot-runner-molding-systems/

[17] (2008). *Hot runner*. Available: http://en.wikipedia.org/wiki/Hot_runner

[18] (2015). *Plastic Injection Molding Materials and Injection Molding Resins*. Available: http://www.paramountind.com/injection-molding-material.html

[19] (2012). *Resin Selection: Why Material Matters*. Available: http://injectionmolding.blog.quickparts.com/2012/11/26/resin-selection-why-material-matters/

[20] (2014). *Button*. Available: http://en.wikipedia.org/wiki/Button

[21] Autodesk, "Autodesk® Moldflow® Insight Plastics made perfect.," ed: © 2009 Autodesk, Inc., 2009.