International Journal of Scientific Research in Mechanical and Materials Engineering
© 2019 IJSRMME | ISSN : 2457-0435

Determination of Effects of Process Parameters on Injection-Molded Plastic Bucket using Moldflow Analysis

Shubham Saxena, M. S. Khidiya, B. L. Salvi, M. A. Saloda
Department of Mechanical Engineering, College of Technology and Engineering, Udaipur, Rajasthan, India

Article Info
Volume 4, Issue 6
Page Number: 01-06
Publication Issue :
November-December-2020

Article History
Accepted : 10 Nov 2020
Published : 20 Nov 2020

ABSTRACT

Plastic injection molding is a revolutionary technique for manufacturing plastic products. Shrinkage and plastic fill in an injection molding process for the application in thin-wall products were addressed in this work. In this paper, Moldflow analysis was done on an injection-molded bucket using Polypropylene (PP) material. CAD model of the plastic bucket was designed in Solidworks and simulated under different molding conditions in Moldflow Adviser to check plastic flow behaviour. Also effects of injection molding process conditions on shrinkage and plastic fill were determined using Moldflow analysis. It was concluded from the study that Moldflow was a significant tool to virtually understand the plastic behaviour inside the mold cavity. It was found that to minimize shrinkage, the melt temperature and mold temperature should be low and the injection pressure and fill time should be high. It was found that the plastic fill was increased with the decrease in melt temperature and mold temperature and the increase in injection pressure and fill time.

Keywords - Injection molding, CAD, Shrinkage, Plastic fill, Moldflow Adviser.

I. INTRODUCTION

Plastic injection molding is one of the most common manufacturing techniques used in cyclic mass production of plastic products. Injection molding process involves injection of melted plastic with pressure into a mold to form it to the cavity shape. Injection molding has three sections viz. injection unit, mold and clamping unit. Injection molding with thin wall products has captured the interest of researchers in this area. Shrinkage and plastic fill were identified as crucial quality characteristics in injection molding from the different studies. Plastic fill is the volume of plastic filled at the end of process [1]. Shrinkage is a dimensional defect related to deformation in injection molded part [2]. Several researches have been done in the field of simulation of injection molding process. Hussin *et al.* (2012) on process optimization of injection molding by reducing the warpage. They concluded that melt temperature contributes the most in minimizing warpage. Altan (2010) determined packing pressure for PP and melt temperature for PS were important parameters for obtaining less shrinkage in the injection-molded part using Taguchi method. Kurt *et al.* (2009) investigated shrinkage and cyclicity of an injection molded ABS part. They concluded that an increase in cavity pressure and mold temperature decreased shrinkage in x and y direction and cyclicity error in part significantly. Tutar *et al.* (2010) simulated single cavity and multi cavity part on Moldflow software for different flow conditions. The mass flow rate was significant parameter for polymer flow front positions in their study. CAE (Computer-Aided Engineering) technique is used to simulate the product virtually before it is manufactured [3]. Many commercial CAE softwares are available for simulation of injection molding process as reported by several researchers [4-7].

In this work, effects of melt temperature, mold temperature, injection pressure and fill time were determined on shrinkage and plastic fill of an injection-molded bucket with the help of Moldflow analysis. Based on the simulation results, effect of variation in process parameter on injection molding process was plotted.

II. METHOD

2.1 Material

The CAD modelling of the 3 D bucket was prepared in Solidworks. It has uniform wall thickness of 2 mm. Autodesk Moldflow Adviser was taken as CAE tool to perform simulation experiments on injection molded bucket. The detailed diagram of the plastic bucket is presented below in figure 1. The thermoplastic material used in study was polypropylene (PP). It is a semi-crystalline material having density of 0.9 g/cm^3 and melt flow index (230 °C/2.16 kg) of 12 g/10 min. Mold material was taken as steel P 20.

The CAD modelling of the 3 D bucket was prepared in Solidworks. It has uniform wall thickness of 2 mm. Autodesk Moldflow Adviser was taken as CAE tool to perform simulation experiments on injection molded bucket. The detailed diagram of the plastic bucket is presented below in figure 1. The thermoplastic material used in study was polypropylene (PP). It is a semi-crystalline material having density of 0.9 g/cm^3 and melt flow index (230 °C/2.16 kg) of 12 g/10 min. Mold material was taken as steel P 20.

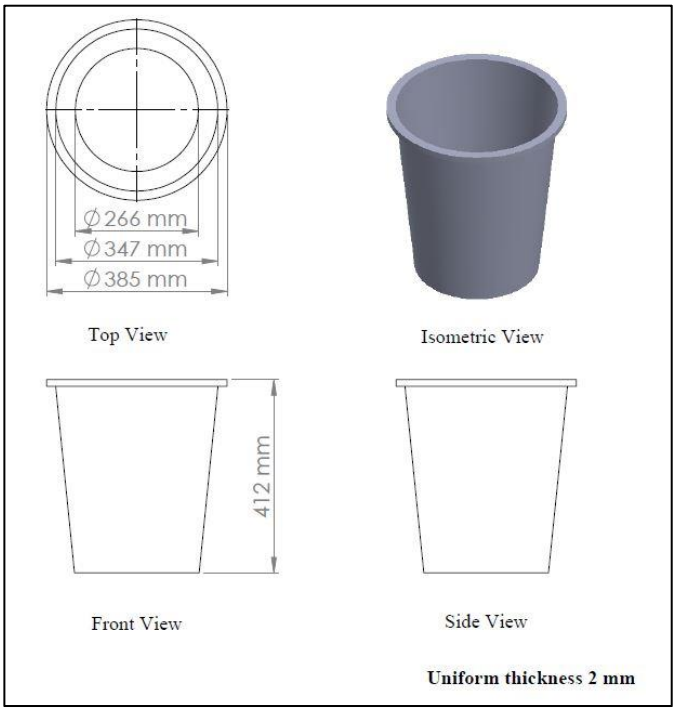


Fig. 1: Preparation of the CAD model

1.1. Measurements

Flow behaviour and quality characteristics of polypropylene (PP) material were analyzed by importing CAD model of the bucket in Moldflow Adviser. To determine shrinkage below mentioned equations 1, 2 and 3 were used.

$$\Delta D_u = \frac{D_u - d_u}{D_u} \times 100\% \qquad ......(1)$$

$$\Delta D_l = \frac{D_l - d_l}{D_l} \times 100\% \qquad ......(2)$$

$$\Delta H = \frac{H - h}{H} \times 100\% \qquad ......(3)$$

Where ΔD_u, ΔD_l and ΔH are percentage shrinkage, D_u, D_l and H are initial while d_u, d_l and h are final upper diameter, lower diameter and height of bucket respectively.

The determination of plastic fill was done by using equation 4 below,

Plastic fill = Weight of final product / Density of material ......(4)

1.2. Boundary conditions and process setup

A single cavity mold was used in the study. The model meshed into total of 13412 CAD triangles and 6708 nodes. The Dual Domain module was selected on Moldflow. Single injection location was selected after gate location analysis. In figure 2, that the cavity inside the mold was presented and bounded by single boundary condition i.e. injection location. Fill and pack analysis was applied to determine the flow behaviour of the plastic.

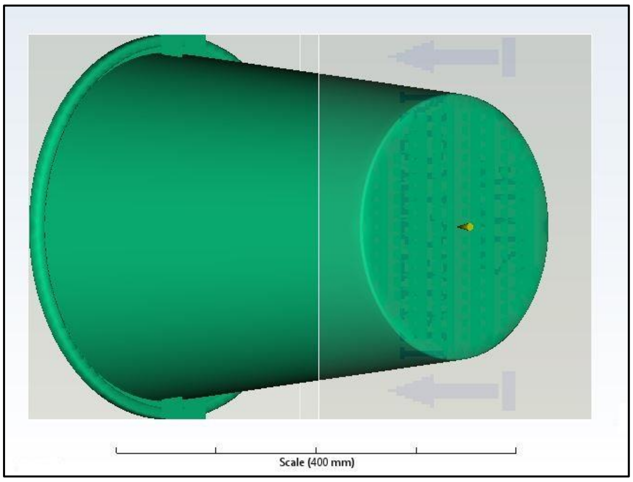
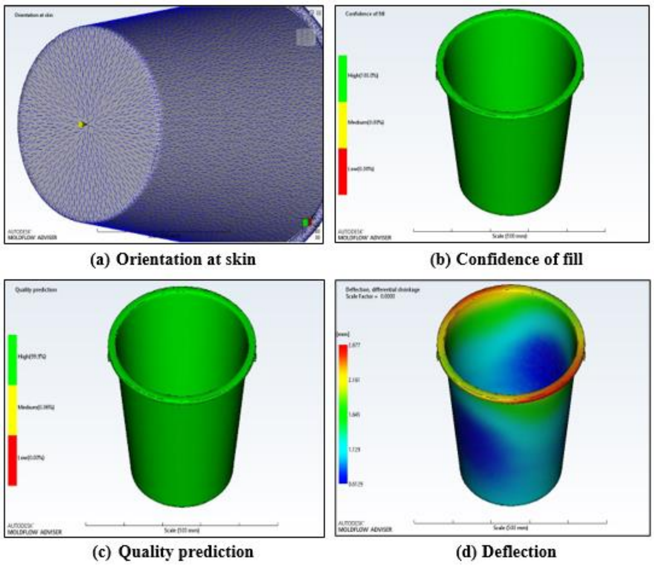


Fig. 2 : Single gate location

The process conditions range for melt temperature, mold temperature, injection pressure and injection or fill time was selected as 190 – 260 °C, 40 – 60 °C, 90 – 180 MPa and 4 – 6 sec respectively. The operating conditions to predict the best flow behaviour was determined. The method of experimentation was to change only one parameter at a time keeping other parameters' value fixed at 250 °C melt temperature, 45 °C mold temperature, 180 MPa injection pressure and 5 sec fill time to study its effect on injection-molded part. For an instance in simulation run, melt temperature was varied at 190 °C, 225 °C and 260 °C while other parameters were mold temperature, injection pressure and fill time, taken as 45 °C, 180 MPa and 5 sec respectively.

III. RESULTS AND DISCUSSION

After analysis, report was generated and studied. In the figure 3, orientation at skin, confidence of fill, quality prediction and deflection results from Moldflow analysis of bucket for the best flow conditions are presented. It helped in the study of the plastic flow behaviour inside mold for a particular condition and predict the effects of variation virtually in injection molding.



(a) Orientation at skin (b) Confidence of fill (c) Quality prediction (d) Deflection

Fig. 3 : Analysis results from Moldflow

It can be determined from orientation at skin result that plastic molecules are aligned in the direction of flow. Thus part strength will be good when impact was applied in the orientation direction. High confidence of fill result showed that the plastic can be easily injected inside the mold cavity. Quality prediction result showed that the part quality and appearance were reliable and acceptable. It helped in emphasizing the area where problems may occur under molding conditions. Deflection result showed the change in overall dimensions of the injection molded bucket. These deflections data were used to determine the final dimensions and shrinkage in the bucket.

In table 1, observation data obtained from Moldflow analysis of injection molding process were presented. In this table shrinkage, dimensions and plastic fill measured from analyzed bucket model on Moldflow were showed.

Table 1: Observation table of Moldflow analysis

Process Parameter		Shrinkage (%)			Part Dimensions (mm)			Plastic Fill (x 10^3 mm^3)
		ΔD_u	ΔD_l	ΔH	Upper Diameter (D_u)	Lower Diameter (D_l)	Height (H)	
Melt Temperature (°C)	190	0.51	0.65	0.70	345.24	264.28	409.12	955.84
	225	0.69	0.69	0.91	344.61	264.17	408.23	941.92
	260	0.76	0.81	1.07	344.35	263.83	407.58	930.75
Mold Temperature (°C)	40	0.61	0.68	0.74	344.89	264.19	408.94	935.75
	60	0.73	0.74	0.84	344.47	264.02	408.55	927.11
	80	0.79	0.85	1.12	344.26	263.74	407.38	917.70
Injection Pressure (MPa)	90	0.75	0.91	1.05	344.39	263.56	407.67	894.72
	135	0.65	0.78	0.98	344.74	263.91	407.88	936.05
	180	0.51	0.71	0.80	345.21	264.11	408.72	961.67
Fill Time (sec)	4	0.68	0.87	1.06	344.65	263.69	407.62	925.61
	5	0.63	0.81	0.88	344.82	263.84	408.37	933.45
	6	0.48	0.66	0.76	345.33	264.24	408.86	940.05

3.1 Effect of melt temperature

In figure 4, the effect of variation in melt temperature on the shrinkage value was showed. Shrinkage for upper diameter, lower diameter and height was increased with the increase in melt temperature of plastic from 190 °C to 260 °C. Contraction of molecules is the cause of shrinkage of plastic inside the mold cavity when provided cooling. When injection molding at high melt temperature, there was more shrinkage. Plastic fill was decreased with the increase in melt temperature from 190 °C to 260 °C. For plastic material at high melt temperature, its viscosity decreased. The filled material's specific volume is increased. Hence for high melt temperature, part of low weight was obtained which in turn decrease the plastic fill inside the mold cavity. Plastic fill was comparatively high for low melt temperature.

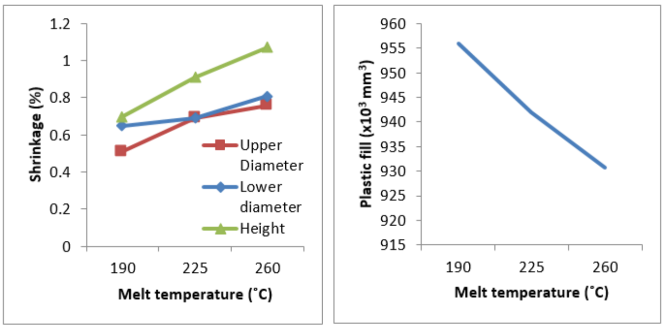


Fig. 4 : Effect of melt temperature

3.2 Effect of mold temperature

From figure 5, it can be determined that with the increase in mold temperature of plastic from 40 °C to 80 °C, the value of shrinkage for upper diameter, lower diameter and height increases. Low mold temperature provided more heat transfer from hot polymer to mold walls than high mold temperature on cooling. Solidification is fast which in turn reduces the shrinkage value for less mold temperature. Shrinkage obtained was more for high mold temperature. It was observed that with the increase in mold temperature from 40 °C to 80 °C there was decrease in plastic fill. At low mold temperature comparatively more plastic can be filled than at high mold temperature due to high density of polymer during the solidification on mold walls.

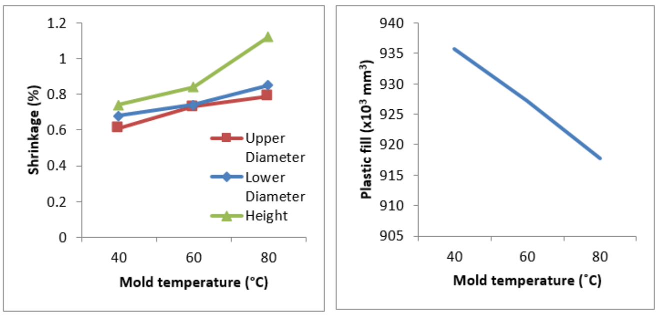


Fig. 5 : Effect of mold temperature

3.3. Effect of injection pressure

In figure 6, it is presented that increasing the injection pressure from 90 MPa to 180 MPa, the value of shrinkage for upper diameter, lower diameter and height was decreased. With high injection pressure, shrinkage can be compensated with the flow advancement of hot polymer inside mold. It was observed that with the increase in injection pressure there was increase in plastic fill. High injection pressure contributed to more density of plastic filled inside mold. Therefore more plastic fill was observed when high injection pressure was provided.

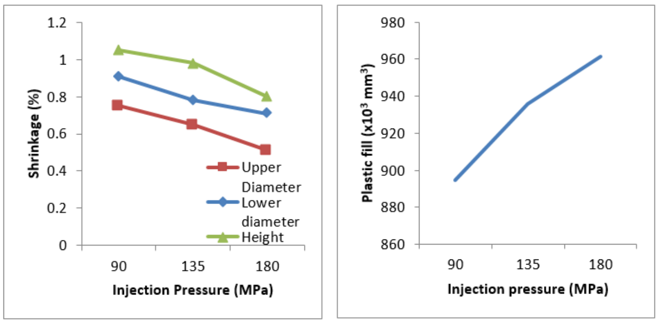


Fig. 6 : Effect of injection pressure

3.4. Effect of injection or fill time

In Figure 7, it can be determined that with the increase in fill time from 4 sec to 6 sec, the value of shrinkage for upper diameter, lower diameter and height was decreased. Time for solidification of plastic is more at high fill time. Due to this shrinkage was less for high fill time than low fill time. It was noted that with the increase in fill time, plastic fill was increased. At high fill time, due to shrinkage during the plastic flow, the plastic fill was more inside cavity than at low fill time.

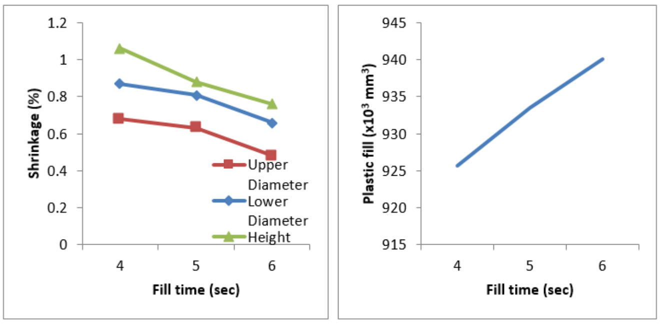


Fig. 7 : Effect of fill time

IV. CONCLUSION

In the present work, below mentioned conclusions were drawn:

1. The study was done to analyze a CAD model of injection-molded bucket on Moldflow Adviser and to observe the plastic behaviour inside the mold cavity. Orientation at skin result showed that the molecules were fairly aligned in the direction of flow. Confidence of fill was high which showed that the plastic was fully injected inside the cavity. The quality prediction was high for part showing that the appearance of part was acceptable. The deviation in part dimensions were measured with the help of deflection results.
2. Plastic processing conditions were simulated using Moldflow analysis that helped to virtually observe the plastic flow inside the mold. It was an effective tool in the study to predict the defects and eliminate them before going to production hence reducing the cost of experimentation.
3. The effect of melt temperature, mold temperature, injection pressure and injection time on the shrinkage percentage and plastic fill were presented by graphs. It was found that less shrinkage in plastic was obtained for 190 °C melt temperature, 40 °C mold temperature, 180 MPa injection pressure and 6 sec fill time. It was determined that plastic fill was high for 190 °C melt temperature, 40 °C mold temperature, 180 MPa injection pressure and 6 sec fill time.

V. REFERENCES

[1]. Moldflow design guide (2006), Moldflow Corporation.

[2]. Jain, K. and Das, S. R. 2016. Modelling and Optimisation of Shrinkage in Plastic Injection Molding of Polypropylene. 4th International Conference on Production and Industrial Engineering, CPIE.

[3]. Liao, S. J., Chang, D. Y., Chen, H. J., Tsou, L. S., Ho, J. R., Yau, H. T., Hseih, W. H., Wang, J. T. and Su, Y. C. 2004. Optimal Process Conditions of Shrinkage Warpage of Thin-Wall Parts. Polymer Engineering And Science 44: 917-928.

[4]. Mehat, N. M. and Kamaruddin, S. 2011. Investigating the effects of injection molding parameters on the mechanical properties of recycled plastic parts using the Taguchi method. Materials and Manufacturing Processes 26: 202-209.

[5]. Hussin Radhwan, Saad, R. M., Hussin, R. and Dawi, M. S. I. M. 2012. An Optimization of Plastic Injection Molding Parameters Using Taguchi Optimisation Method. Asian Transactions on Engineering 2: 75-80.

[6]. Miranda, D. and Nogueira, A. 2019. Simulation of an injection process using a CAE tool: assessment of operational conditions and mold design on the process efficiency. Materials Research 22: 1-16.

[7]. Mohamed, O. A., Masood, S.H. and Saifullah, A. 2013. A Simulation Study of Conformal Cooling Channels in Plastic Injection Molding. International Journal of Engineering Research 2: 344-348.

[8]. Kurt, M., Kamber, O. S., Kaynak, Y., Atakok, G. and Girit, O. 2009. Experimental investigation of plastic injection molding: assessment of the effects of cavity pressure and mold temperature on the quality of the final products. Materials and Design 30: 3217-3224.

[9]. Tutar, M. and Karakus, A. 2010. Computational Study of Effect of Governing Parameters on a Polymer Injection Molding Process for Single Cavity and Multi Cavity Mold Systems. Journal of Manufacturing Science And Engineering 132: 1-12.

[10]. Altan, Mirigul 2010. Reducing shrinkage in injection moldings via the Taguchi, ANOVA and neural network methods. Materials and Design 31: 599-604.

Cite this article as :

Shubham Saxena, M. S. Khidiya, B. L. Salvi, M. A. Saloda, "Determination of Effects of Process Parameters on Injection-Molded Plastic Bucket using Moldflow Analysis",International Journal of Scientific Research in Mechanical and Materials Engineering (IJSRMME), ISSN : 2457-0435, Volume 4 Issue 6, pp. 01-06, November-December 2020.
URL : http://ijsrmme.com/IJSRMME20458