Modeling and flow analysis of pure nylon polymer for injection molding process

2016 IOP Conf. Ser.: Mater. Sci. Eng. 114 012043

(http://iopscience.iop.org/1757-899X/114/1/012043)

Modeling and flow analysis of pure nylon polymer for injection molding process

D M Nuruzzaman[1], N Kusaseh[2], S Basri[2], A N Oumer[3] and Z Hamedon[2]

[1,2] Faculty of Manufacturing Engineering, University Malaysia Pahang, Malaysia
[3] Faculty of Mechanical Engineering, University Malaysia Pahang, Malaysia

E-mail: dewan052005@yahoo.com

Abstract. In the production of complex plastic parts, injection molding is one of the most popular industrial processes. This paper addresses the modeling and analysis of the flow process of the nylon (polyamide) polymer for injection molding process. To determine the best molding conditions, a series of simulations are carried out using Autodesk Moldflow Insight software and the processing parameters are adjusted. This mold filling commercial software simulates the cavity filling pattern along with temperature and pressure distributions in the mold cavity. In the modeling, during the plastics flow inside the mold cavity, different flow parameters such as fill time, pressure, temperature, shear rate and warp at different locations in the cavity are analyzed. Overall, this Moldflow is able to perform a relatively sophisticated analysis of the flow process of pure nylon. Thus the prediction of the filling of a mold cavity is very important and it becomes useful before a nylon plastic part to be manufactured.

1. Introduction

In recent years, using the high speed digital computer and commercial simulation software. the mathematical analysis of the flow processes of the polymer is gaining popularity. Computational polymer processing is very important for the information of different flow parameters. The detailed and important up-to-date information about the rheology and polymer processing computations have been discussed by several authors [1-5]. Polymers can be processed by different methods such as blow molding, film blowing, fiber spinning, extrusion, thermoforming, compression molding, injection molding etc. Among these methods, extrusion and injection molding are the most important polymer processing operations [6]. Injection molding involves the steps: heating and melting of polymer, inject the melted polymer to mold cavity, forming the melted polymer into the required dimensions/shape, packing, cooling and solidification and finally, the ejection of the molded product [1]. Under the same processing conditions, the effects of velocity and pressure on the melt front advancements of HDPE and PP were investigated by both experimentally and using an injection molding analysis program [7]. In the analyses, six different injection velocities and five different injection pressures were used. It was found that experimentally obtained melt front profiles are almost similar to the theoretical results, but the filling times are different.

[1] Corresponding author

Using an injection molding software package, the process of filling a mold by polypropylene resin has been simulated [8]. In the simulation, the parameters include an injection pressure of 9.5 MPa, mold temperature of 50◦C and a resin melt temperature of 230◦C. The analysis result showed that predicted complete cavity filling time is 0.71 sec. Shrinkage rate of a polystyrene product during injection molding process is investigated [9]. The obtained results show that shrinkage rate distribution depends on the cooling system and the cooling process of the plastic product is much dependent on the position of the cooling channels. A comparison between theoretical and experimental temperature measurements has been made during the injection molding cycle [10]. The obtained results reveal that during the molding cycle, there are several rise in the temperature events at the surface of the molded part. The difference between theoretical calculations and experimental measurements allowed to explain a physical interpretation of the temperature event of the semi-crystalline polymer (PP) molding.

In this study, flow analysis of pure nylon polymer has been carried out using commercial Autodesk Moldflow Insight software. Different flow parameters such as fill time, pressure distribution, temperature at flow front etc. are analyzed. Gating suitability is also analyzed for the injection of the melt nylon polymer.

2. Flow analysis and processing technique

A plastic injection molded part can be successfully produced when a well-designed concept, right choice of the material and appropriate manufacturing process are brought together. The flow analysis of pure nylon is very important to identify the suitable process flow of the injection process. The flow process of the nylon polymer for the injection molding is modelled and analyzed using Autodesk Moldflow Insight software. In the modeling, during the plastics flow inside the mold cavity, different flow parameters such as pressure, temperature, fill time etc. are analyzed. Three different designs of gate location for dog bone shaped nylon specimen that are designed by using Catia V5 software. These three designs are considered to undergo a Moldflow analysis and the best gate location for a good dog bone shaped specimen produced is selected. Figure 1 shows three different dog bone shaped modelings of nylon specimen using Catia V5.

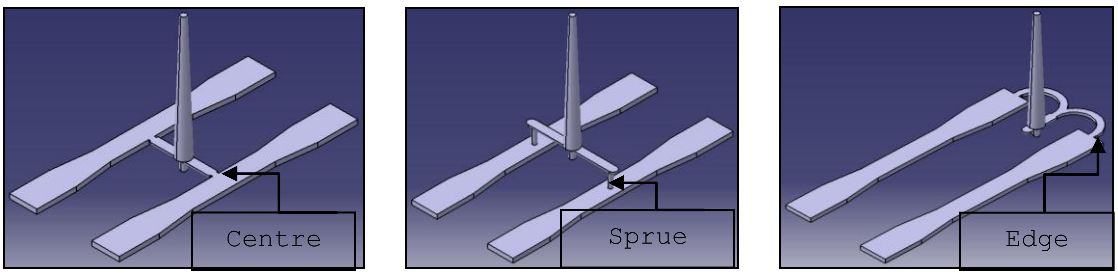


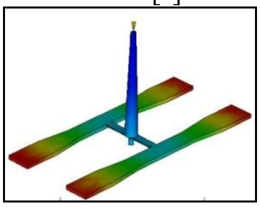
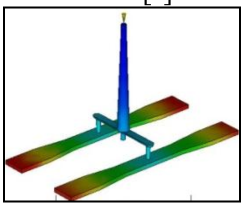
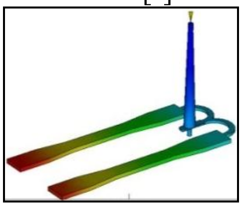
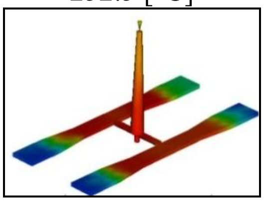
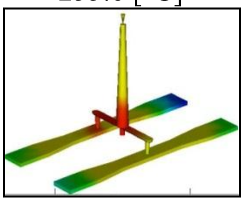
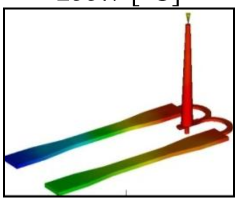
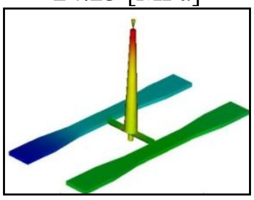
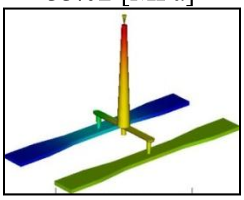
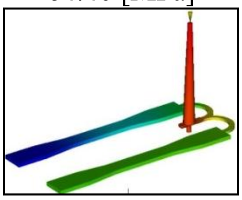
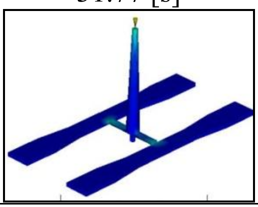
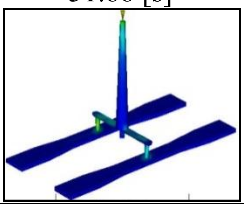
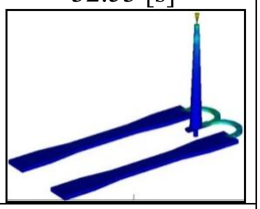
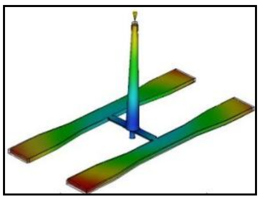
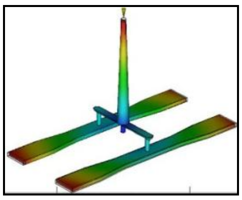
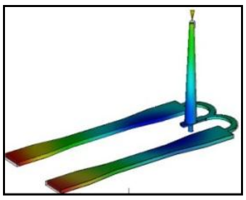
Figure 1: Three different 3D modelings of dog bone shaped nylon specimen

Table 1 shows a summary of different Moldflow analyses for three different type of gate locations. The Moldflow analysis shows different values or results for different type of gate locations used. For the best gate location, the edge gate is chosen because, fill time figure shows a uniform parallel direction or parallel orientation across the whole width even though the fill time for edge gate is longer than centre gate or sprue gate. Moreover, considering the molded product shape and size, edge gate is selected. This gate is located on the edge of the part and is the best suited for flat part like the dog bone shaped nylon specimen with thickness 3.2 mm only. Centre gate is not chosen because this type of gate location would reduce strength of the plastic part. Sprue gate is also not chosen because of the direct injection pressure to the molded product, residual strain would occur which in turn may affect

IOP Conf. Series: Materials Science and Engineering **114** (2016) 012043 doi:10.1088/1757-899X/114/1/012043

the appearance of the molded plastic part as the gate location is on the upper surface of the specimen. The gating suitability result rates each place on the model for its suitability for an injection location. Moldflow analysis shows the gating suitability in figure 2. The most suitable areas, colored blue, are rated as best, and the least suitable areas of the model, colored red, are rated as worst. The blue color shows that it is the best location to inject the molten plastic to the mold cavity. A rating of blue (best) does not necessarily mean that the part can be filled from this location. This is to be checked by running a full analysis.

Table 1: Tabulated Moldflow Analysis

No.	**Moldflow Analysis**	**Centre Gate**	**Sprue Gate**	**Edge Gate**
1	Fill Time	1.802 [s] 	1.915 [s] 	2.634 [s] 
2	Temperature at Flow Front	252.5 [°C] 	255.6 [°C] 	255.7 [°C] 
3	Pressure at End of Fill	24.23 [MPa] 	33.02 [MPa] 	64.40 [MPa] 
4	Shear Rate, Max.	31.77 [s] 	31.88 [s] 	32.53 [s] 
5	Warp			

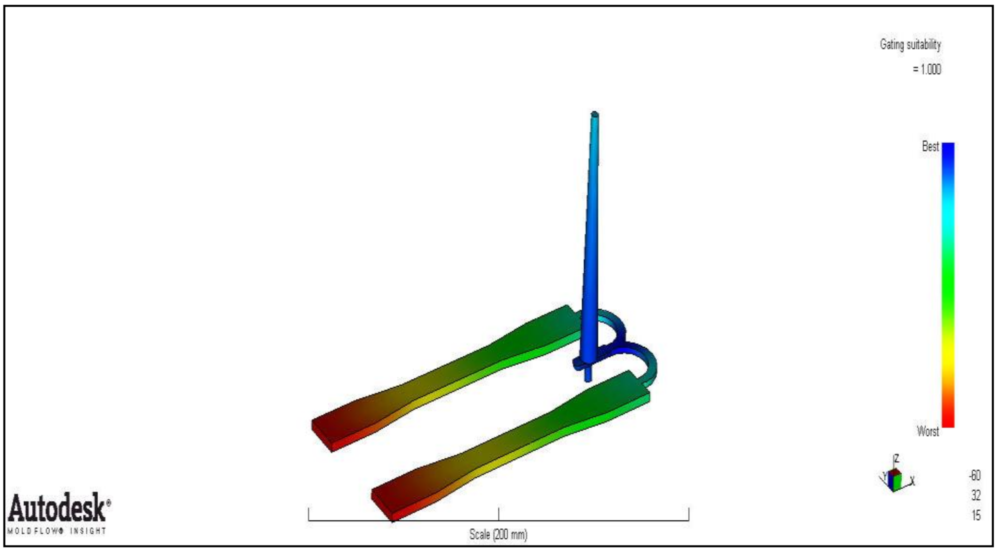


Figure 2. Moldflow analysis for the gating suitability

3. Results and discussion

Figure 3 shows the filling time of the mold cavity for the dog bone shaped nylon polymer specimen. In injection molding process, mold filling analysis plays a significant role in order to determine the quality of molded nylon specimen. Molding defects like short shot can be prevented by accurate prediction of filling time. Short shot is the part of the model that does not fill. A short shot has no color and appears as translucent. In the analysis, changes are made in the processing parameters to ensure proper filling of the cavity and short shot in the molded plastic part is prevented. In the modeling, the obtained results indicate the filling time for the plastic material flow path through sections using contour connecting the areas filling at the same time. The contours are shown in different colors from blue to indicate that the first areas to fill, to red, to fill to show the last region. For the edge gate location, the fill time displays proper filling within the mold cavity and the results show that total filling time is 2.634 seconds (red colored).

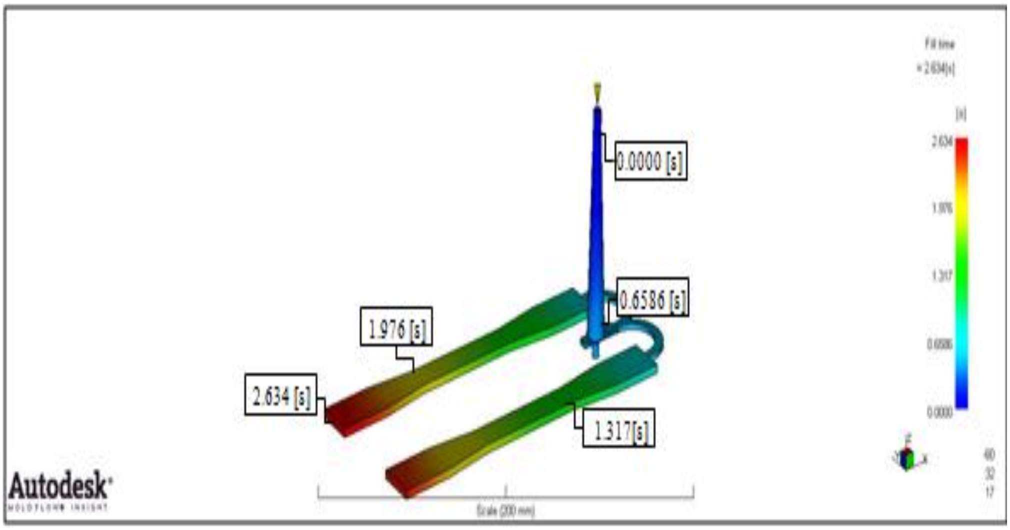


Figure 3. Moldflow analysis for the filling time

Figure 4 shows the temperature distribution of the molded nylon part that is generated from the Moldflow analysis. The temperature at flow front uses different colors which represent the material temperature at different points during the filling process. Fill analysis shows the temperature of the nylon polymer when the flow front reaches at a certain point in the middle of the plastic cross-section. The obtained results show the changes in the temperature of the flow front during mold filling. The region of highest temperature uses red color while the region of lowest temperature uses blue color. During the filling phase, the processing parameters are adjusted so that the flow front temperature variation is minimized to under 2-5^{o}C to prevent the material degradation and surface defects. Results show that temperature of the flow front of melt nylon polymer varies from 255.7^{o}C to 245.9^{o}C during the mold filling process. It is understood that during the filling phase, the temperature drop is about 2.5^{o}C at different stages of filling.

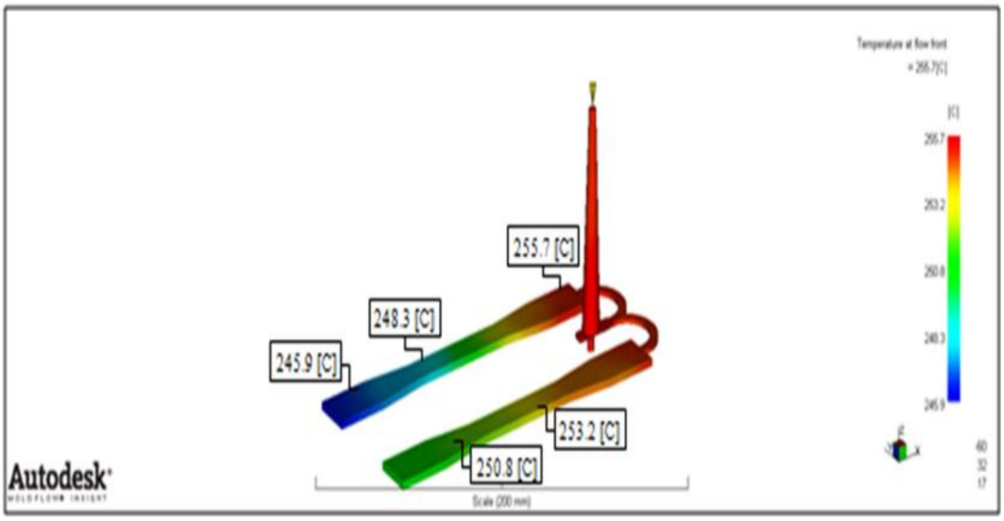


Figure 4. Moldflow analysis for the temperature at flow front

Figure 5 shows the pressure distribution in the mold cavity at the instant when the cavity is completely filled with nylon polymer. At the start of the mold filling process, pressure is zero throughout the mold cavity and pressure starts to increase at a specific location when melt front reaches that location. During the filling process, the force that pushes the polymer melt to flow is due to the difference in pressure from one location to another location. During the mold filling process, as the melt front moves past, the pressure continues to increase. From the figure, it can be noted that at the nylon injection locations, the maximum pressure exists as 64.4 MPa. On the other hand, during the filling process, minimum pressure exists at the melt front, and at the furthest point, pressure is zero at the end of filling process.

Figure 6 shows the shear rate, maximum result of the nylon polymer i.e. the maximum shear rate at a given node and from the fill analysis, this result is generated. In general, the shear rate is a measure of how quickly nylon layers are slided past each other. For very high shear rate, polymer chains break and consequently the material degrades. As shown in the figure, the shear rate at a maximum state is 4056.2 (1/s) at time = 32.53 seconds.

Figure 7 shows the Moldflow warp analysis results of nylon polymer. Warpage in the injection molded nylon part occurs due to the variations in shrinkage at different locations of the molded part. To achieve the dimensional accuracy in the molded nylon part, low uniform shrinkage is desirable at different locations within the part. From the warp analysis results it is understood that warp of the molded part can be minimized and controlled by selecting the appropriate design parameters and processing conditions.

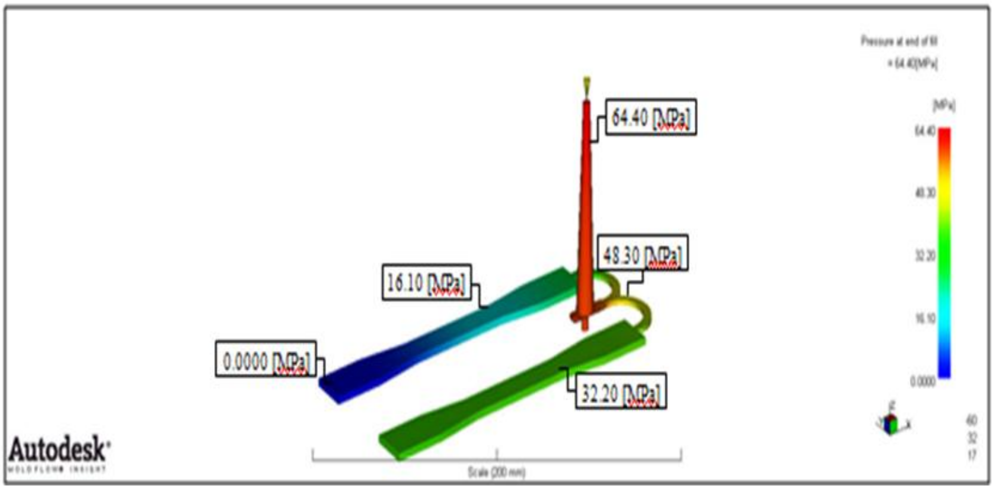


Figure 5. Moldflow analysis for the pressure distribution

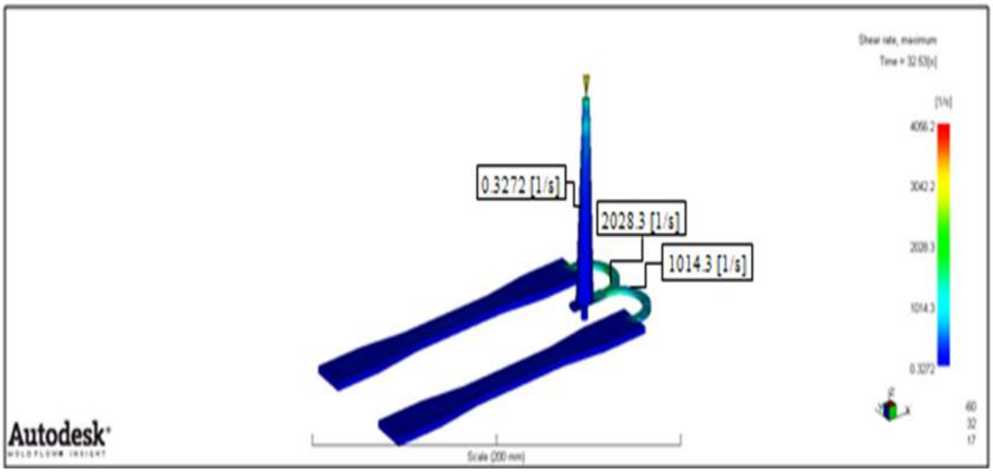


Figure 6. Moldflow analysis for the shear rate, maxumum

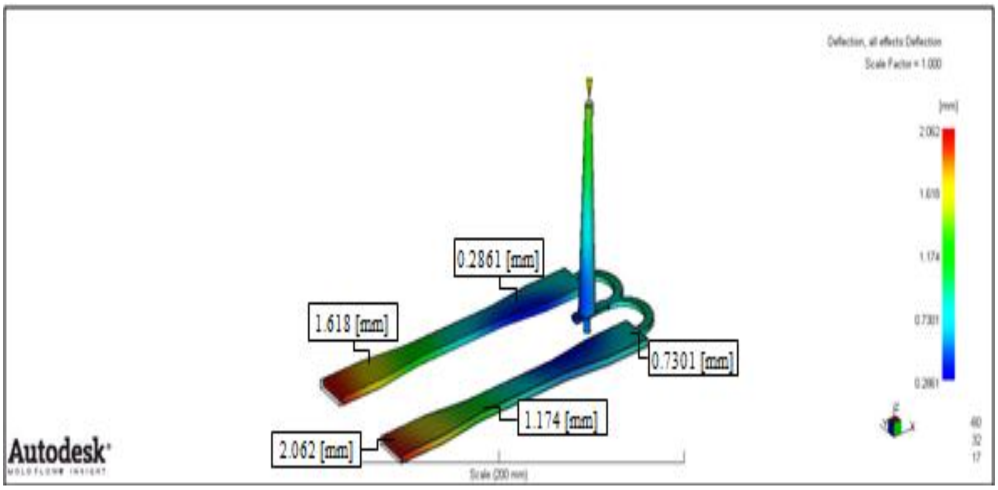


Figure 7. Moldflow warp analysis

4. Conclusion

A successful plastic injection molded part is a combination of a well-designed concept, the right material selection and an appropriate manufacturing process. For the injection molding process, different flow parameters of nylon polymer resin are analysed using Moldflow software. The position of best gate is generated by the Best Gate Location analysis for its suitability for an injection location and ensures the dog bone shaped nylon polymer specimen can be filled completely and successfully. The obtained simulation results show promising results. From filling analysis, necessary data such as filling time, temperature at flow front, pressure distribution etc. are obtained. The predicted results are found to be reliable to describe the effect of processing conditions on the injection molded component and can be used for a real time manufacturing process.

5. References

[1] Tadmor Z and Gogos C G 2006 Principles of polymer processing, SPE monograph series, 2nd edition, John Wiley & Sons, Inc. New York
[2] Han C D 2007 Rheology and processing of polymers, Vol. 1, Polymer rheology, Oxford University Press, Oxford
[3] Han C D 2007 Rheology and processing of polymers, Vol. 2, Polymer processing, Oxford University Press, Oxford
[4] Kennedy P K 2008 Practical and scientific aspects of injection molding simulation PhD Thesis, Technical University of Eindhoven (TUE), The Netherlands
[5] Owens R G and Phillips T N 2002 Computational rheology, Imperial College Press, London, pp. 19-47
[6] Vlachopoulos, J. and Strutt, D. 2003 Overview: Polymer Processing. Mat. Sci. Tech., 19, 1153–1160.
[7] Ozdemir A, Uluer O and Guldas A 2004 Flow front advancement of molten thermoplastic materials during filling stage of a mold cavity, *Polym. Test.* **23** 957–66
[8] Tatara R A, Sievers R M and Hierzer V 2006 Modeling the injection molding processing of a polypropylene closure having an integral hinge *J Mater Process Tech* **176** 200-4
[9] Hassan H, Regnier N, Pujos C, Arquis E and Defaye G 2010 Modeling the effect of cooling system on the shrinkage and temperature of the polymer by injection molding *Appl. Therm. Eng.* **30** 1547-57
[10] Nicolazo C, Sarda A, Vachot P, Mousseau P and Deterre R 2010 Change on temperature at the surface of injection moulded parts *J Mater Process Tech* **210** 233-7