VOL. 10, NO. 15, AUGUST 2015 ISSN 1819-6608
ARPN Journal of Engineering and Applied Sciences

www.arpnjournals.com

COMPARISON BETWEEN STAR AND LINEAR RUNNER LAYOUT OF FAMILY PLASTIC INJECTION MOLD

Mohd Amran Md Ali, Noorfa Idayu, Rosidah Jaafar, Mohd Hadzley, Yuhazri Yaakob, Abdul Rahim Samsudin and Mohd Rizal Salleh
Precision Machining Group (PMG), Faculty of Manufacturing Engineering, Universiti Teknikal Malaysia Melaka, Malaysia
E-Mail: mohdamran@utem.edu.my

ABSTRACT

Comparison of various responses such as fill time, deflection, volumetric shrinkage and residual stress between runner linear layout and runner star layout using mold flow simulation software in family plastic injection molding was studied. The plastic parts such as tensile specimen, impact specimen, flexural specimen and hardness specimen were designed using CATIA version 5. Then, designed plastic parts in CATIA software were imported into Moldflow software to transform the plastic specimen parts from solid form to mesh form. Feeding system such as sprue, runner and gate including water cooling system was designed inside the mold. Determination of the type of injection molding machine and the type of plastic material in the settings was taking under cool + fill + pack + warp analysis. It is found that runner linear layout produces lower filling time as compared to runner star layout and deflection on runner linear layout was lower than runner star layout. Then, shrinkage value in runner star layout exceeded runner linear layout. In addition, in cavity residual stress, distribution stress on the plastic part surface of the star runner layout shows high stress. Thus, it shows that runner linear layout in family plastic injection molding was the most suitable layout for this family injection mold.

Keywords: Simulation, star runner layout, linear runner layout.

INTRODUCTION

Harper [1] stated that injection molding is called as versatile process which can produce parts as small as a fraction of a gram and as large as 150 kg. In injection molding process, molten plastic is forced or injected into a mold and cooled until the melt solidified. When the part is cooled sufficiently, the mold is opened. After that the part is ejected from the mold, and the mold is closed to repeat the cycle. In addition, 90 percent of injection molding occurs with thermoplastic resins and it permits mass-production, high precision, and three-dimensional virtual net shape manufacturing of plastic parts. According to Harper [2] thermoplastics soften and melt during heating which allows them to be shaped using plastics processing equipment. The purpose of an injection mold is to give the shape of the part, distribute the polymer melt to the cavities through a runner system, cool the part, and eject the part. During the injection molding cycle, the polymer flows from the nozzle on the injection unit through the sprue, and then to the runners, which distribute the melt to each of the cavities. The entrance to the cavity is called the gate and is usually small so that the runner system can be easily removed from the parts.

FLOW SIMULATION

Introduction of Flow Simulation

In 2006, Walsh [3] found that Moldflow Mold Advisor is an example of a program that will analyze plastic parts and their molds to predict problems such as trapped air, short shots, weld line and etc. When the user has specified one or more gate locations, the software then simulates the cavity filling with plastic. Based on the simulation of how the mold fills, the software then will warn the designer of potential problems with warping, sink marks, or trapped air. If a potential problem exists, the designer can use the analysis to redesign the mold. Gheorghe *et al.* [4] have performed a numerical analysis using the Autodesk Moldflow Insight for process optimization of the injection of a “surgical micro stitch” in a mold with four nests, in order to obtain a high quality device. The simulation of the filling process allows for improving quality of the final device and significant reduction of costs through shortening time for design and production. In addition, injection time of the piece has a significant influence on solidification of the piece. Maier [5] stated that family’s molds show big pressure differences between cavity areas. The expected result was a wide tolerance dimension in size of parts, density and mechanical properties. Moldflow software provides easy to redesign mold layout to ensure equal pressure drops to each cavity. The result was a minimum material usage through the elimination of over pack in some cavity area. Furthermore, alternative sprue runner and gate system can be assessing by examine the flow patterns. Unbalance flow conditions can cause problems unless corrected were done. High stresses or local over packing of the mold cavity area lead to distorted and warping on the end parts.

Amran *et al.* [6] found that fast injection causes shear heating of the melt, thereby requiring the longer cooling times that facilitate relaxation and crystallization. Increased packing of the mold will reduce shrinkage, but this is limited by the gate freeze-off time. Molds with unbalanced filling will also exhibit over packing and under packing. This creates nonuniform shrinkage in the part. Imihetri *et al.* [7] stated that the result of filling time represented the behavior of the molten polymer at regular periods. Plastic flow inside the mold was simulated using a program that calculated a flow front that grows from interconnecting nodes at each element, starting at the node of injection. The cycle repeated until the flow front had fully expanded to fill the last node. One of the goals in selecting gating injection locations was to ensure that all flow paths in the cavity fill at the same time (flow paths balancing). This prevented over packing along the flow paths, which can otherwise fill first another area. Yin *et al.* [8] said five key process parameters were selected as the design variables in the mathematical model for the optimization of injection molding process using neutral network and genetic algorithm. These are mold temperature (Tmold), melt temperature (Tmelt), packing pressure (Pp), packing time (tp) as well as the cooling time (tc). The upper and lower bounds of the process parameters are set based on the recommended values provided by Moldflow software. More than that the neutral network and genetic algorithm optimization are used in wider application [9,10]. Reddy *et al.* [11] said during production, quality problems of the plastic parts such as warpage, shrinkage, weld and meld lines, flow mark, flash, sink mark and void are affected from manufacturing process conditions which include the melt temperature, mold temperature, injection pressure, injection velocity, injection time, packing pressure, packing time, cooling time, cooling temperature and etc. They also added by considering the more number of input process parameters such as injection velocity, injection time, runner types, gate location together with the process parameters of the mold temperature, melt temperature, packing pressure, packing time and cooling time to study the effect of warpage of injection molded parts. Choi and Im [12] stated during the filling stage of injection molding, high pressure is applied to force polymer melt into the mold. In this process, a thin layer solidifies at the contact with the mold surface. However, this thin layer has little effects on the shrinkage and warpage of the parts, because the pressure of the filling stage is much lower than that of the packing stage. The shrinkage of both materials decreases with increasing melt temperature. As melt temperature increases, the gate freezing is delayed such that more materials can be added to decrease the amount of shrinkage. In general, increase of mold temperature delays the gate freezing time which decreases shrinkage while increasing thermal shrinkage after ejection. Farshi *et al.* [13] said warpage deflection and volumetric shrinkage were considered as defects, minimizing both of them was a useful task in manufacturing processes. Warpage means that warping deflection of the part in injection molding due to the non-uniform contraction of different points and geometric shrinkage was the overall contraction of the part when it cooled. Minimizing them resulted in better product quality. Geometrical shrinkage was often adjusted from a coefficient of contraction in mold designs design. Excessive volumetric shrinkage can cause changes on volume that can produce out of dimensional tolerance in the end product. Lastly, it can be said that minimization of deflection warpage in the process of injection molding with compromise to control shrinkage which result in minimum cycle time and less residual stress and optimize for production processing. Rahman *et al.* [14] said that undergoing cooling process in mold where heat was transferred off the part surface effectively and eventually form solidification layer. But the situation was different for the portion under the solidified layer, rates of heat transfer drop drastically across the part thickness. Any unbalance cooling usually causes higher warpage deflection and volumetric shrinkage occur. Shrinkage and warpage were closely related to cooling circuit design in early stage mold design. Higher volumetric shrinkage required extra cooling designs to ensure the heat equally transferred off from cavity area. Small variation of cooling time in a plastic part may cause small warpage problem. In fact, warpage of a plastic part also depends on the percentage of the frozen layer in the plastic part and partly depends on the frozen time. Kurtaran *et al.* [15] found that warpage is inversely proportional with mold temperature, melt temperature, packing pressure and packing pressure time but directly proportional with cooling time at least for the problem of interest. Amran *et al.* [16] found the most important quality problems is warpage. Warpage, is a distortion of the shape of the final injection-molded item, is caused by differential shrinkage; that is, if one area or direction of the article undergoes a different degree of shrinkage than another area or direction, the part will warp. Dang [17] said the objective functions such as warpage, shrinkage, or residual stress are determined to undergo the schematic procedure for optimizing injection process parameters in conjunction with direct simulation-based optimization. The designer identifies the design variables such as melt temperature (Ti), mold temperature (Tm), fill time (ti), packing time (tp), and packing pressure (Pp) as well as constraints. The constraints are usually the range of design variables and some boundary conditions related to the specification of the molding machine. Kwiatkowski *et al.* [18] found a higher mold temperature causes easier cavity filling and a decrease in the temperature gradient between the polymer and mold wall causes a decrease in residual stresses in the part. For lower mold temperature values, an increase in residual stresses of about 5% was observed. Azaman *et al.* [19] found the effect of different cooling times that occurs at the centre of the surface of the thin-walled parts from 10 to 50 s on the residual stresses showed no obvious change. Douven *et al.* [20] stated there were some issues that affect the quality of plastic parts in molding process. Some of the issues were the filling pattern of the mold. The assessment of a mold design such as gate locations, expected of weld location and the vents that necessary to release air entrapped in the cavity area. The locations of the cooling circuit have a great effect on the cycling time, on the temperature of the cavity area, and consequently on the residual stresses and the warpage deflection. When too much material was forced into the mold then consequently over-packing occurs, resulting in problems during ejection the plastic parts from the cavity area. The molded plastic parts have residual stresses, mainly due to differential volumetric shrinkage. Further, warpage deflection happen when the volumetric shrinkage was different over the plastic parts. This can causes by unbalance of cooling or by molecular orientation during frozen process.

EXPERIMENTATION

CATIA V5 was used to draw four different parts layout or called as family mold that consists of tensile test specimen, hardness test specimen, impact test specimen and flexural test specimen. Autodesk Simulation Moldflow® Insight software was used for injection molding simulation analysis. Figure-1 shows the four specimens were created in CATIA software. They were imported to Moldflow environment and meshed with triangular elements. Table-1 shows the model details for runner linear layout and runner star layout.

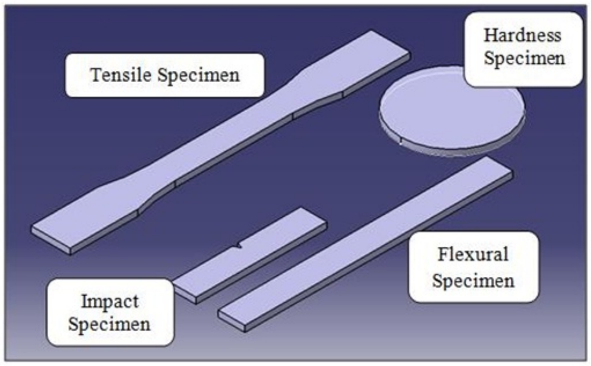

Figure-1. Family mold layout.

Table-1. Model details for runner linear layout and runner star layout.

Model Details	Runner Linear Layout	Runner Star Layout
Mesh match percentage	99.9%	99.9%
Reciprocal mesh match percentage	99.9%	99.9%
Total number of elements	16822	16795
Aspect ratio of triangle elements (Average)	1.4753	1.4753
Aspect ratio of triangle elements (Maximum)	6.8404	6.8404
Aspect ratio of triangle elements (Minimum)	1.1579	1.1579
Volume to be filled	28.60 cm^3	25.18 cm^3
Total projected area	85.35 cm^2	78.09 cm^2

The simulation was started by using runner linear layout as shown in Figure-2 (a) and followed by runner star layout as shown in Figure-2 (b).

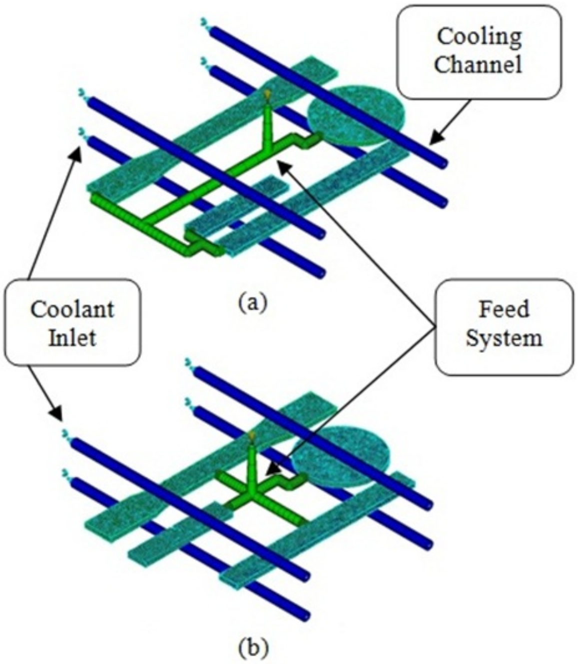

Figure-2. Family mold design by using (a) runner linear layout and (b) runner star layout.

The injection molding machine used is a default setting injection molding machine in the software. Specifications of injection-molding machine are shown in Table-2. Material used is polypropylene (3131 MU7) and its property is shown in Table-3.

Table-2. Specifications of injection-molding machine.

Parameters	Values
Maximum machine injection pressure (MPa)	180
Maximum machine injection rate (cm^3/s)	5000
Maximum machine clamp force (tonne)	7000

Table-3. Polypropylene (3131 MU7) material properties.

Material properties	Values
Melt Mass-flow Rate (g/10 min)	11
Elastic Modulus (MPa)	1340
Shear Modulus (MPa)	481
Poisson ratio	0.4

The processing parameters for both meshed models are shown in Table-4. The processing parameters were specified by the software to use as preliminary inputs for the analysis. The simulation for both models was performed by using set analysis called as (Cool + Fill + Pack + Warp). The result about fill time, deflection, volumetric shrinkage at ejection and in-cavity residual stress along the first principal direction were analyzed.

Table-4. Processing parameters for both meshed models.

Parameters	Runner Linear Layout	Runner Star Layout
Recommended Mold Temperature (^{o}C)	31.67	42.78
Recommended Melt Temperature (^{o}C)	210	203.33
Recommended Injection Time (s)	0.6963	0.3887

RESULTS AND DISCUSSION

Fill Time

The results of fill time simulation for both runner design show that the linear layout required 0.7477 s to fill all parts and the star layout required 1.935 s to fill all parts as shown in Figure-3. The difference in fill time which is approximately 1.1873 shows that runner linear layout fills in at slightly faster time compared to runner star layout.

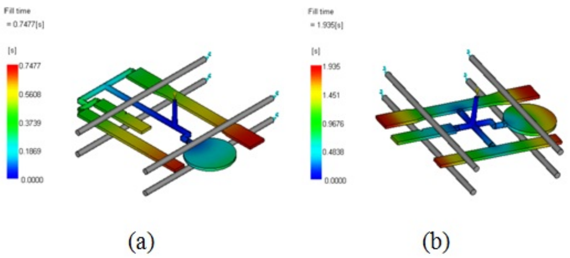

Figure-3. Fill time for (a) runner linear layout, (b) runner star layout.

According to Imihezri *et al.* [7], the result of fill time represented the behavior of the molten plastic at regular period. Molten plastic flow inside the mold is simulated using a program that calculated a flow front that grows from interconnecting nodes at each element, starting at the gating injection node. The cycle repeated until the flow front had fully expanded to fill the last node. The purpose of selection plastic injection gate locations was to ensure all flow paths in the cavity fill area at the same time to obtain a balanced flow path. This is importance to avoid over packing along the flow paths. From the analysis result, it can be seen that in Figure-3 (a), hardness and impact specimens flow path finish before the other such as flexural and tensile specimens. In the simulation, if each flow path ends with red colors, all of the paths are finish at the same time. In Figure-3 (b), hardness and impact specimens flow path finish first before flexural and tensile specimen. Over packing can cause warpage, high part weight and non-uniform density distribution throughout the part.

Deflection

Figure-4 shows the deflection or distribution ranges of warpage in three components (x, y, and z) for both models. In Figure-4 (a), the range of deflection is 0.0177-0.9215 mm, which is lower than 0.0175-0.9574 of Figure-4 (b). Sections in red had the highest warpage defects.

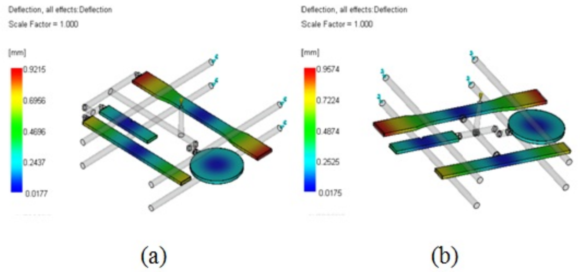

Figure-4. Deflection analysis for (a) runner linear layout, (b) runner star layout.

Choi and Im [12], found that after demolding, the injection molded parts undergo shrinkage and warpage caused by residual stresses and temperature change. Doven *et al.* [20] stated that the location of the cooling channels has a great effect on the warpage. Warpage occurs when the shrinkage is different over the product. This may be caused by inhomogeneous cooling or by frozen-in molecular orientation.

Volumetric Shrinkage at Ejection

In Figure-5, volumetric shrinkage at ejection of runner star layout has high percentage about 11.72% compared to 10.94% of runner linear layout. The volumetric shrinkage at ejection result shows the volumetric shrinkage for each area expressed as a percent of the original modeled volume. Volumetric shrinkage should be uniform across the whole part to reduce warpage.

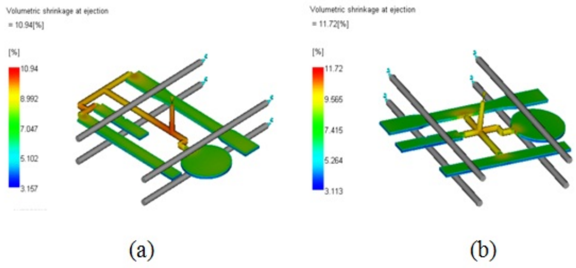

Figure-5. Volumetric shrinkage at ejection analysis for (a) runner linear layout, (b) runner star layout.

According to Mohd *et al.* [21] shrinkage and warpage were closely related to the design of cooling channels in a particular mold. Higher volumetric shrinkage required extra cooling designs in order to ensure the heat was equally released. Farshi *et al.* [13] found that volumetric shrinkage was the overall contraction of the plastic part when it was cooled. Minimizing it, can improve plastic parts quality. In addition, extra shrinkage may cause volumetric changes. Consequently, can produce out of tolerance dimensions in the final plastic parts. Localized areas of high shrinkage can result in internal voids or sink marks when the plastic part frozen.

In-Cavity Residual Stress in First Principal Direction

Residual stresses in the part can be created as a result of shear stresses generated during mold filling or packing. Positive value on the plot indicates tension and at this state the part is still under constraint within the mold. From Figure-6, both models show positive value and over packing did not exist inside both models. In Figure-6 (b) residual stresses is distributed to almost all area of the part compared to Figure-6 (a).

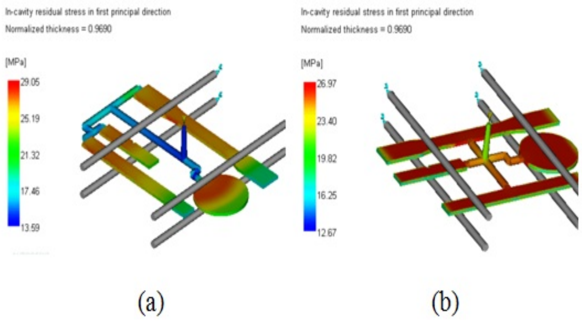

Figure-6: In- cavity residual stress in first principal direction analysis for (a) runner linear layout, (b) runner star layout.

In 2001, Kansal *et al.* [22] found that residual stress in injection molded part arise due to the non-isothermal flow of the polymer in the mold cavity during filling and packing. The difference in cooling rate of the polymer near the boundary and inside the cavity also can contribute to residual stress. Therefore, uniform cooling is required to minimize the residual stresses. Kwiatkowski *et al.* [18] stated that a higher mold temperature causes easier cavity filling and a decrease in the temperature gradient between the polymer and mold wall causes a decrease in residual stresses in the part. For lower mold temperature values, an increase in residual stresses was observed. Further, Suzuki *et al.* [23], stated that unbalance of flow instability due to different type of plastic materials viscosity had resulted high level of onset shear stress. According to Azdast *et al.* [24], due to the residual stress induced during post filling stage, an amount of shrinkage was released at part ejection.

CONCLUSIONS

In this study, the effect of arrangement runner layout between runner star layout and runner linear layout towards fill time, deflection, volumetric shrinkage at ejection, and in-cavity residual stress in a plastic injection mold is studied. It is found that runner linear layout gives better results for all the responses studied. This is because the fastest filling time can be achieved using linear runner layout. While, the deflection result from this study shows that linear layout also has given smaller impact on deflection than the star runner layout. A significant reduction of the impact of volumetric shrinkage is also achieved from the results using linear layout compared to runner star layout. Reduction effect of in-cavity residual stress on the surface of the plastic part is also produces from runner linear layout. Therefore, in the future work, the researcher should use runner linear layout in selection of runner for producing these products, i.e., tensile, hardness, impact and flexural specimen, using the family injection mold.

ACKNOWLEDGEMENTS

The authors would like to thank Faculty of Manufacturing Engineering, Universiti Teknikal Malaysia Melaka (UTeM) for providing facilities for this research to be conducted successfully. This research is funded by the Ministry of Higher Education through Fundamental Research Grant Scheme (FRGS/1/2014/TK01/FKP/02/F00224).

REFERENCE

[1] Harper C. 2000. Modern Plastics Handbook. New York: McGraw-Hill.

[2] Harper C. 2002. Handbook of Plastics, Elastomers, and Composites (4th ed.). New York: McGraw-Hill.

[3] Walsh R.A. 2006. McGraw-Hill Machining and Metalworking Handbook (3rd ed.). New York: McGraw-Hill.

[4] Gheorghe O., Florin T., Vlad G. and Gabriel D. 2014. Optimization of micro injection molding of polymeric medical devices using software tools. Procedia Engineering, pp. 340-346.

[5] Maier C. 1985. The Moldflow system of computer aided polymer flow analysis. Materials & Design, pp. 239-243.

[6] Amran M., Hadzley M., Amri S., Izamshah R., Hassan A., Samsi S. and Shahir K. 2010. Optimization of gate, runner and sprue in two-plate family plastic injection mold. AIP Conference Proceedings, 1217, pp. 309–313.

[7] Imihezri S., Sapuan S., Sulaiman S., Hamdan M., Zainuddin E., Osman M. and Rahman M. 2006. Mold flow and component design analysis of polymeric based composite automotive clutch pedals. Journal of Materials Processing Technology, pp. 358-365.

[8] Yin F., Mao H. and Hua L. 2011. A hybrid of back propagation neural network and genetic algorithm for

optimization of injection molding process parameters. Materials & Design, pp. 3457-3464.

[9] Valdez F., Melin P. and Castillo O. 2014. Modular neural networks architecture optimization with a new nature inspired method using a fuzzy combination of particle swarm optimization and genetic algorithms. Information Sciences, Vol. 270, pp. 143-153.

[10] Shang Y. and Bouffanais R. 2014. Influence of the number of topologically interacting neighbors on swarm dynamics, Scientific Reports, Vol 4. p. 4184

[11] Reddy B., Kumar J., Reddy V. and Padmanabhan G. 2009. Application of soft computing for the prediction of warpage of plastic injection molded parts. Journal of Engineering Science and Technology, pp. 56-62.

[12] Choi D. and Im Y. 1999. Prediction of shrinkage and warpage in consideration of residual stress in integrated simulation of injection molding. Composite Structures, pp. 655-665.

[13] Farshi B., Gheshmi S. and Miandoabchi E. 2011. Optimization of injection molding process parameters using sequential simplex algorithm. Materials & Design, pp. 414-423.

[14] Rahman W., Sin L. and Rahmat A. 2008. Injection molding simulation analysis of natural fiber composite window frame. Journal of Materials Processing Technology, pp. 22-30.

[15] Kurtaran H., Ozcelik B. and Erzurumlu T. 2005. Warpage optimization of a bus ceiling lamp base using neural network model and genetic algorithm. Journal of Materials Processing Technology, pp. 314-319.

[16] Amran M., Salmah S., Mohd Z., Raja I., Mohd H., Sivarao S., Mohd S. and Mohd A. 2014. The effect of pressure on warpage of dumbbell plastic part in injection molding machine. Advanced Materials Research, pp. 61–66

[17] Dang X. 2014. General frameworks for optimization of plastic injection molding process parameters. Simulation Modelling Practice and Theory, pp. 15-27.

[18] Kwiatkowski D., Gnatowski A. and Nabiałek J. 2011. Numerical analysis of residual stresses and deformation of injection molded parts manufactured from polymeric composite with different processing conditions. Kompozyty, pp. 294-298.

[19] Azaman M., Sapuan S., Sulaiman S., Zainudin E. and Khalina A. 2014. Numerical simulation analysis of the in-cavity residual stress distribution of lignocellulosic (wood) polymer composites used in shallow thin-walled parts formed by the injection molding process. Materials & Design, pp. 381-386.

[20] Douven L., Baaijens F. and Meijer H. 1995. The computation of properties of injection-molded products. Progress in Polymer Science, pp. 403-457.

[21] Mohd A., Siti S., Raja I., Mohd S., Mohd A., Effendi M., Mohd S., Sivaraos Zolkarnain M., Umar A., Mohd K. and Zulkeflee A. 2015. Warpage analysis of different number cooling channels for dumbbell plastic part in injection molding. Applied Mechanics and Materials, pp. 8-11.

[22] Kansal G., Rao P. and Atreya S. 2001. Study: Temperature and residual stress in an injection molded gear. Journal of Materials Processing Technology, pp. 328-337.

[23] Suzuki M., Md Ali, M.A., Okamoto K., Taniike T., Terano M., Yamaguchi M. 2009. Effect of stereoregularity of polypropylene on flow instability in capillary extrusion. Advance in Polymer Technology, pp. 185-191.

[24] Azdast T., Behravesh A., Mazaheri K. and Darvishi M. 2008. Numerical simulation and experimental validation of residual stress induced constrained shrinkage of injection molded parts. Polimery, pp. 304-31.