JOURNAL OF MODERN MANUFACTURING SYSTEMS AND TECHNOLOGY (JMMST)
e-ISSN: 2636-9575
VOL. 5, ISSUE 2, 1 – 8
DOI: https://doi.org/10.15282/jmmst.v5i2.6652

ORIGINAL ARTICLE

The impact of injection mold's gate modification on the quality of PC/ABS blend using CAE simulation

F. Hentati[1], N. Masmoudi[2]

[1] National Engineering School of Tunis ENIT, Applied Mechanics and Engineering Laboratory (LR-MAI), Tunis, 1002, Tunisia.
[2] National Engineering School of of Sfax ENIS, Electromechanical Systems Laboratory (LASEM), Sfax, 3038, Tunisia.

ABSTRACT – PC/ABS blend is widely used in producing the automotive components due to its thermal and mechanical properties. However, variations in the process condition can have an effect on its properties. Finding the optimal injection process parameters plays a very important role in controlling the quality and the mechanical properties of the molded parts. A material temperature of 260 °C, an injection pressure of 50 bar, a holding time of 8 s and a mold temperature of 60 °C are the optimal combination parameters yielded a better shear stress and no injection defects. These results were found in our previous work and confirmed using the Taguchi method and SolidWorks Plastics. Moreover, factors affecting on the part's quality can be essentially the injection parameters or the mold cavity. This paper presents a simple way to investigate the impact of a modification in the mold's gate size system on injected PC/ABS parts' quality. Optimal parameters are tested via SolidWorks simulation to explore the defects occurring following this modification. This study outcomes show some defects with the new mold's gate size like weld lines and the residual stresses, also a high inlet pressure achieved compared to the actual mold's gate.

ARTICLE HISTORY
Received: 12th July 2021
Revised: 2nd August 2021
Accepted: 4th August 2021

KEYWORDS
Injection Molding
PC/ABS parts
Mold's gate
Defects
CAE simulation

INTRODUCTION

Nowadays, plastic injection molding plays a critical role in the industrial environment and involves a wide variety of applications like packaging, aerospace, construction and automotive [1- 4]. Mold design, material characteristics and process conditions are the quality evaluation criteria of the injected parts [5]. Injection mold shape and the process parameters specify the final quality of the injected part. Mold shape is considering an important factor in producing the accurate injected part. Hence, the mold's characteristics such as gate selection, its position and size must be taken seriously for the conception of the plastic injection mold [7-8]. These characteristics must be carefully designed to ensure a final part without defects and to define significant mechanical properties [8-10].

Researchers have been previously investigated the impact of the mold's design on the final quality of molded parts. For example, Hassan et al [11] investigated the effect of the gate location on the cooling of injected molded part. They carried out a full three dimensional time-dependent analysis for a mold with cuboids-shape cavity containing two different thicknesses. Furthermore, Vijaykumar et al [12] used a component named to as flow reducer that used to understand design review and mold flow analysis. They applied the software Autodesk, Mold Flow for the conception and the simulation of both the product and the mold. In parallel, Mehdi Moayyedian et al [13] suggested an innovative design of new cross sectional shape of runner system in the plastic injection molding. They looked to find an adequate conception which enable to reduce the cycle time and to scrap easier ejection of runner system from mold tools. Besides, Titomanlio et al. [14] examined the shrinkage and the residual stress distributions of the final products and their effect on the injection mold. In addition, Angstadt and Coulter [15] investigated the link between cavity pressure and part quality product in the injection molding process. They considered that the use of cavity pressure as significant potential indicator of the final part quality. In other recent work, Hsu [16] applied the MoldFlow Software to conduct a simulation analysis on aspheric lenses. Also, they adopted the Taguchi Method to optimize the injection molding parameters on the shrinkage of aspheric lenses. Results demonstrated that gate size, holding pressure and filling duration have a considerable impact on the volumetric shrinkage rate. Similarly, Rajalingam et al [17] used Taguchi approach to explore the process parameters generating the shrinkage defect on a plastic cell phone shell component. Optimal molding process parameters are identified using the Design of Experimental (DOE). They consider mold temperature, injection pressure and screw rotation speed as the process parameters in their research study. Also, Jianga et al [18] conducted a study for simulation of injection molding to control volume finite element method. They used local flow information to control time steps for both flow and thermal simulation. Finally, Koya et al [19] examined gate location, diameter and defects through the filling and packing phase of knob that used to set/tune the time for a wash in Washing machine via MoldFlow software and hyper mesh.

*CORRESPONDING AUTHOR | F. Hentati | ✉ fatma.hentati1@gmail.com

From the literature presented above, we can conclude the importance of the numerical simulations in science and engineering for the analysis of mold flowing; and the Taguchi method in solving optimization conditions.

The aim of this work is to study the impact of the mold's gate size modification on the final quality of the injected PC/ABS parts by integrating two approaches : the Design of Experiment (DOE) using Taguchi and the Computed Aided Engineering (CAE) via SolidWorks Plastics analysis.

The DOE has been implemented to select optimal process parameters that could result in a better quality product and shear stresses. Solid Works applied this optimal combination in order to predict the type of defects that could happen following the mold's gate change.

EXPERIMENTAL STUDY

Material and Mold Design

The material elaborated is the commercial grade of PC/ABS blend labeled T45 PG. It is widely used in many industrials application such as packaging, aerospace, construction and particularly in the automotive field.

According to the BAYER® company, supplier of the PC/ABS blend, the designation of the T45 PG means that it is theoretically composed of 55% ABS and 45% PC. The properties of PC/ABS used in this study are summerized in Table 1 [20].

Table 1. Mechanical and thermal characteristics of T45 PG.

Properties	Method	T45 PG
MVR (260°C/5kg)	ISO 1133	12 cm^3/10min
Melt viscosity(1000s^{-1}/260°C)	ISO 11443	200 Pa.s
Vicat softening temperature	ISO 306	112°C
Tensile strength (50mm/min)	ISO 527	49 MPa
Izod notched impact strength (23°C)°	ISO 180	40 kJ/m^2

The injected molded PC/ABS parts were produced in the SKG factory. It is a local company specialized in the injection and electroplating of the PC/ABS automotive parts, established in Sfax, Tunisia.

To produce suitable specimens (Figure. 1), a mold was designed (Figure. 2) and fabricated in the SKG company. As illustrated in Figure. 2(a), three cavity imprints were elaborated. During the plastic injection molding process, the tolerance of the molded part has been considered.

The originality of using this plastic injection mold is in producing three mechanical test specimens: Tensile (E_{tr}), Dynamic tensile (Et_{rd}) and torsion (E_{tor}) samples as illustrated in Figure 1 (a). The tensile test specimens conforms to the standard ISO 527-2. The dynamic tensile samples (E_{trd}) conforms to the standard ISO 8256. Mechanical specimens dimensions are given in Figure. 1(b). Each specimen has a specific geometry different from the others. This difference will be marked in the molten polymer flows at the end of the filling phase.

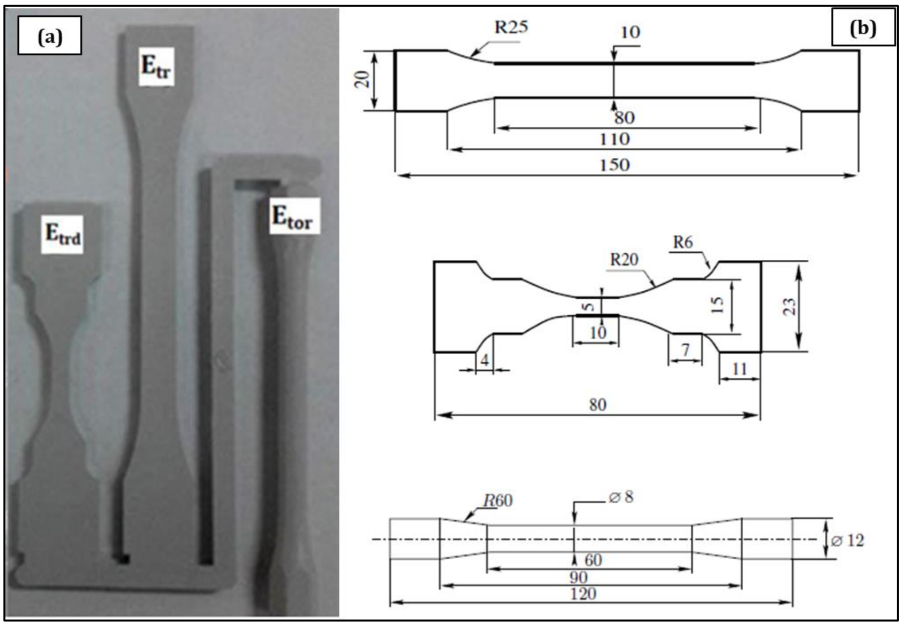

Figure 1. The obtained specimens (a) and their dimensions(b) in mm.

Figure 2(a) illustrates the plastic injection mold considered in this investigation. The actual gate size of the injection mold is depicted in Figure. 2(b). The gate or also called the orifice of molten material into the cavities of the injection mold is one of the most important design variables in mold design. It will affect filling time, temperature and pressure distributions. The edge gate is the type considered in this investigation. This type is in agreement with J.P. Beaumont [21] who reported that the edge gate is considered the most basic that has the rectangular cross section whose main advantage is for use in multi cavity molds.

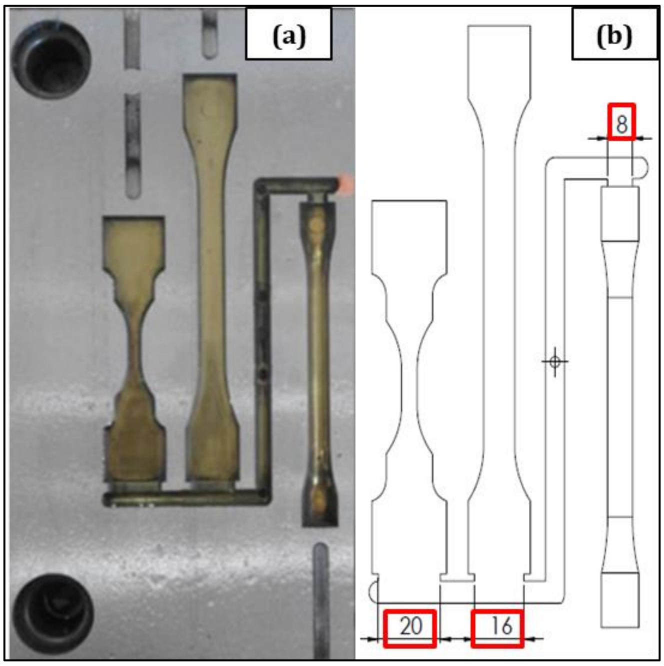

Figure 2. The injection mold (a) and the actual gate size in mm (b).

Numerical Simulation Protocol

Optimal process parameters found via Taguchi approach using the actual mold's gate size are depicted in Table 2 [20]. This optimal combination yielded best shear stress with no injection defects.

Table 2. Optimal process parameters.

Process parameters	Values
Material temperature	260 °C
Injection pressure	50 bar
Holding Time	8 sec
Mold temperature	60 °C

Figure 3 presents molded PC/ABS part elaborated according to the optimal process combination. It confirmed effectively that there is no defect observed on the surface of the injected part.

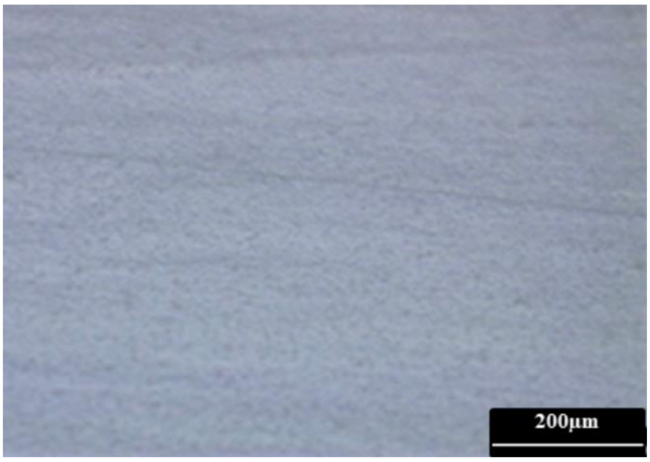

Figure 3. Flawless surface for the optimal injection parameters.

Such modification in the mold could affect the way in which the molten polymer flows into the mold cavity. Therefore, a reduction up to 50% in the mold's gate was proposed by the SKG Company to explore the behavior of the material and to find out the defect occurred by applying the same optimal parameters used with the actual gate size.

Our work is in agreement with some scholars like W. Dai et al [22] and D.A.R. Calhoun et al [23] who revealed that the gate dimension should be as small as possible to avoid the occurrence of visible blemishes on the part's final quality and excess material removal. Similarly, Li et al [24] applied a methodology for single gate location optimization for plastic injection mold in order to optimize the warpage via numerical simulation software MPI. In parallel, Shen et al [25] established a new methodology that predicts the optimum gate location of injection molds based on injection molding simulation. Also, M. Moayyedian et al [7,8] who changed the size and geometry of runner and gate system in order to reduce the scrap rate of injected parts.

SIMULATIONS

Gate Modification Analysis

The gate size of the mold is reduced to 50% in term of dimensions from that of the actual gate size. This modification resulted from a slick entering of the molten polymer blend into the cavities by creating flow direction during the injection process.

The mold's characteristics are defined as follow: the injection point Ip and the dimensions of the actual and the proposed gate size change in the Figure. 4 (a) and Figure. 4 (b), respectively.

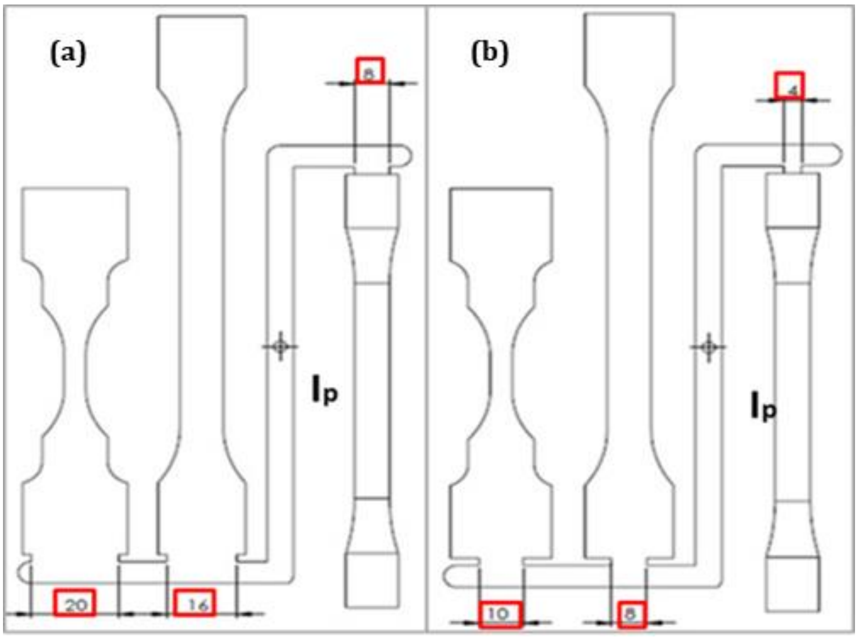

Figure 4. The actual (a) and the proposed (b) gate size modification in mm.

Simulations Results

The optimal injection molding parameters are defined according to the material blend and the injection machine used. Material temperature of 260 °C, injection pressure of 50 bar, mold temperature of 60 °C and holding time of 8 sec are the adequate parameters for this simulation. The type of meshing is cubic. The cycle time, cooling time, internal defects such as residual stresses will be affected by the new gate system dimension.

Ease of filling, filling time analysis, weld and meld lines analysis and injection pressure at the end of filling are the key factors to evaluate the suggested edge gate size.

As depicted in Figure. 5, the ease of filling level is ranging between 0 and 2 with three colors. Red color refers to a bad level of filling, yellow color for the average level and the green color for the best level of filling. Both of the existing and the proposed gate size system reached the most satisfactory level represented by the green color.

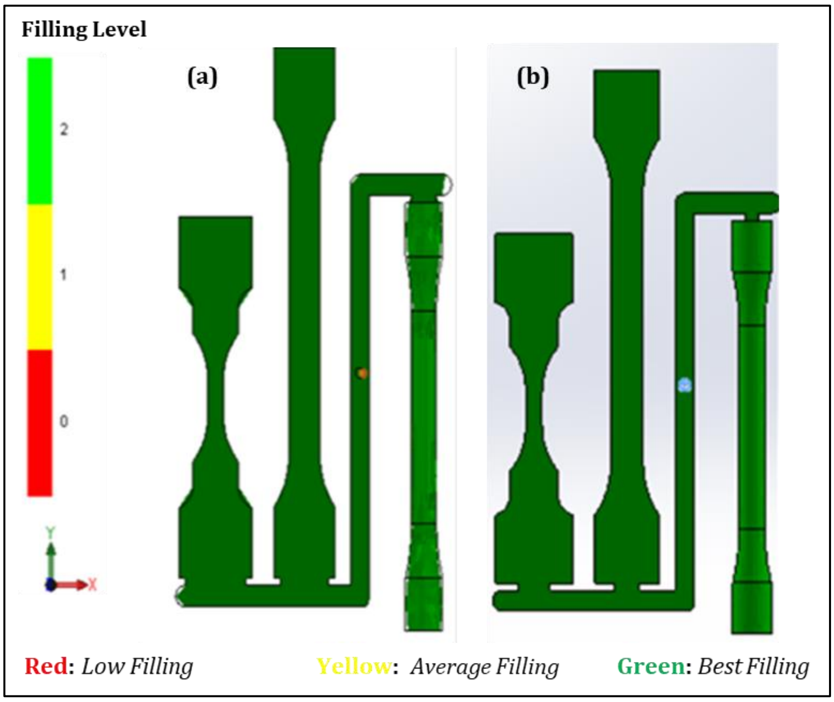

Figure 5. Easy filling of actual (a) and new proposed (b) gate size system.

Another chief factor to verify the efficiency of the new proposed gate system is the filling time. The difference in term of filling time between the existing and the new gate size system is negligible. This finding is proved via the numerical simulation depicted in Figure. 6. We can conclude that both injected part are well filled within an acceptable filling time.

Researchers like Li, Chen et al [26] stated that the short shot defect would happen during the injection molding exactly when there are long flow distances or on thin walls and far from the gate. This situation was sensibly avoided especially that the holding time in this simulation was 8 sec.

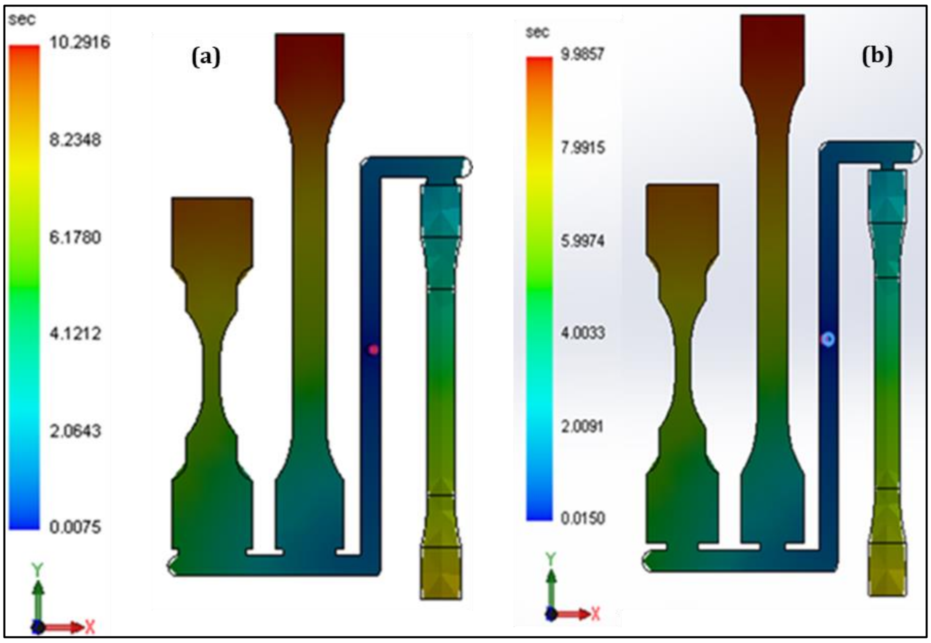

Figure 6. Filling time of actual (a) and new proposed (b) gate size system.

The weld lines defect is observed in the new suggested gate size near to the dynamic tensile and the torsion specimens as illustrated in Figure 7(b), which is not the case for the actual gate system (Figure. 7(a)).

The proposed modification on the gate size system induced the occurrence of this blemish. Weld lines represent a threat in the plastic industry since they cause to weaken or/and break of the injected molded parts. This defect is the result of the thin and solidified layers of the fronts of the molten polymer flow paths meet and fuse, before solidifying again with the rest of the plastic. Therefore, the occurrence of such defect could cause a serious structural and appearance problems of the injected parts [27, 28].

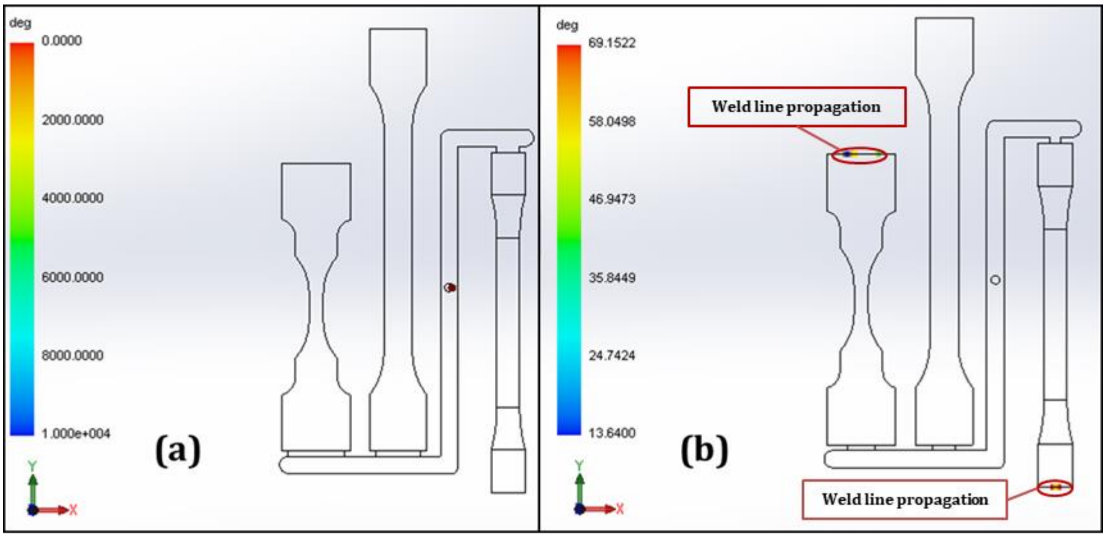

Figure 7. Weld line propagation of actual (a) and new proposed (b) gate size system.

The residual stresses are the stresses lasting inside the molded part under the condition of no external loads. They are solidified inside the part in the molding process. These internal defects are the principal reason of part shrinkage and warpage.

Some scholars investigated the effect of these blemishes. For instance, Östergen [29] defined two types of residual stress; namely, flow-induced residual stress and thermal-induced residual stress. This author and Erzurumlu and Ozcelik [30] agreed that the existence of these defects in injected parts present a serious problem meanwhile their high values may lead to self-released cracking of the mold pieces and their deformation.

As shown in Figure. 8, the residual stresses were spread in all the injected PC/ABS parts. The runner system and the bottom part of the tensile specimen closest to the injection point Ip are the mostly attacked by these internal defects. Besides, residual stresses basically have a great impact on the strength properties of the injected PC/ABS parts. They may remain fixed inside the molded specimens and may cause a major transformation in their dimensions.

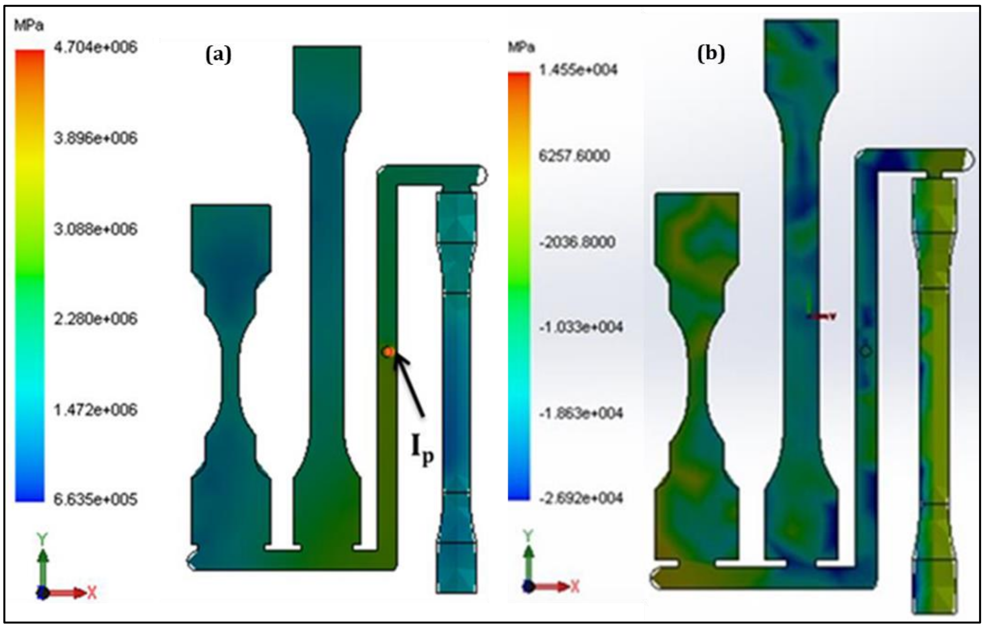

Figure 8. Residual stresses of actual (a) and new proposed (b) gate size system.

The inlet maximum pressure is considered one of the most significant criterions for the evaluation of the part geometry. It is clearly seen from Figure. 9 that injected parts of the existing and the new proposed gate system are filled successfully with an injection pressure of 2.55 and 4.75MPa, respectively.

This pressure is considered high compared to the actual mold's gate size. This finding confirms that the modification of the mold cavity generates certain defects and slows down the filling process and subsequently increases the pressure at the inlet.

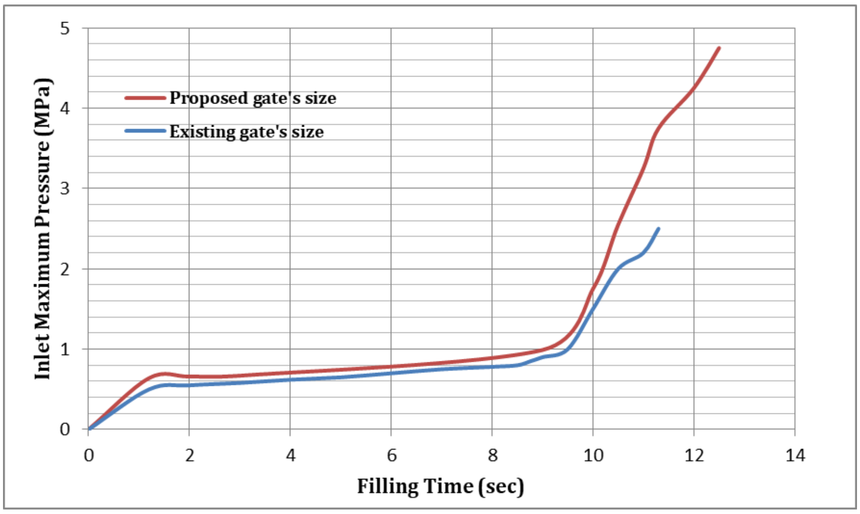

Figure 9. Inlet injection pressure as a function of filling time.

CONCLUSION

Computer Aided Engineering (CAE) via SolidWorks Plastics simulation is successfully used to determine the defects generating following the mold's modification. Optimal injection process parameters including a material temperature of 260 °C, an injection pressure of 50 bar, a holding time of 8 s and a mold temperature of 60 °C confirmed by Taguchi are tested numerically.

Results prove that the modified gate system in comparison with the actual gate has more turbulences of molten polymer flows into the cavities, which leads to the occurrence of some internal and external defects in the final injected parts such as weld lines, residual stresses and high inlet pressure. These blemishes will demean the final quality of the injected PC/ABS part and decline the metallization process. The experimental set up will be conducted as future work to validate the simulation result.

REFERENCES

[1] V. Panneerselvam, F. M. Turan, "Optimization of Process Parameters of Injection Moldings for Plastic Pallets Manufacturing Industry," *Journal of Modern Manufacturing Systems and Technology*., 2, pp. 75-83, 2019.

[2] A. Salimi , M. Subaşı, and L. Buldu, "Prediction of flow length in injection molding for engineering plastics by fuzzy logic under different processing conditions, " *Iranian Polymer Journal.*, 22(1), pp. 33-41, 2013, doi 10.1007/s13726-012-0103-5.

[3] M. Amran, M., Salmah, S., Zaki, "The effect of pressure on warpage of dumbbell plastic part in injection moulding machine," *Advanced Materials Research*., 903, pp. 61-66, 2014, doi:10.4028/www.scientific.net/amr.903.61.

[4] N. R. Subramanian, L.Tingyu, Y. A. Seng, "Optimizing warpage analysis for an optical housing," *Mechatronics*., 15(1), pp. 111-127, 2005, doi.org/10.1016/j.mechatronics.2004.05.004S.

[5] M. Altan, "Reducing shrinkage in injection moldings via the Taguchi, ANOVA and neural network method," *Materials & Design*., 31(1), pp. 599-604, 2010, doi.org/10.1016/j.matdes.2009.06.049.

[6] P.P. Shinde, S. Awati, P. Suryavanshi, "Design of feed system of plastic injection mold for housing retainer," *Journal of Emerging Technologies and Innovative Research (JETIR)*., Vol. 6, March 2019, Issue 3.

[7] M. Moayyedian, K. Abhary, R. Marian, "Elliptical cross sectional shape of runner system in injection mold design," *International Journal of Plastics Technology*., 20(2), pp. 249-264, 2016.

[8] M. Moayyedian, K. Abhary, R. Marian, "Improved gate system for scrap reduction in injection molding processes," *Procedia Manufacturing*., 2, pp. 246-250, 2015.

[9] W. Dai, P. Liu, X.Wang , "An improved mold pin gate and its flow pattern in the cavity," *Journal of Injection Molding Technology*., 6(2), 115, 2002.

[10] P. Jones, "The Mould Design Guide,"Smithers Rapra Technology Limited, 2008.

[11] H. Hassan, N. Regnier, G. Defaye, "A 3D study on the effect of gate location on the cooling of polymer by injection molding," *International Journal of Heat and Fluid Flow*., 30(6), pp. 1218-1229, 2009, doi.org/10.1016/j.ijheatfluidflow.2009.06.005.

[12] V. V. Andhalkar, D. S. Dulange, "Injection molding methods design, optimization, Simulation of plastic flow reducer part by mold flow analysis," *International Research Journal of Engineering and Technology (IRJET) e-ISSN*., vol. 04, June, pp. 1742-1746, 2017.

[13] M. Moayyedian, K. Abhary, R. Marian, "New design feature of mold in injection molding for scrap reduction," *Procedia manufacturing*., 2, pp. 241-245, 2015. doi.org/10.1016/j.promfg.2015.07.042

[14] G. Titomanlio, K. M. B. Jansen, "In-mold shrinkage and stress prediction in injection molding," *Polymer Engineering & Science*., 36(15), pp. 2041-2049, 1996, doi.org/10.1002/pen.10599

[15] D. C. Angstadt, J. P. Coulter, "Cavity pressure and part quality in the injection molding process," *Intell Mater Manuf Lab.*, Lehigh University, Pennsylvania, USA, 1999.

[16] T.T. Hsu, "Study on simulation and optimization in injection molding of plastic aspherical lens," *Master's Thesis, Feng Chia University*., Taiwan, ROC, 2004.

[17] S. Rajalingam *et al.*, "A statistical experimental study on shrinkage of injection-molded part, " *International Journal of Humanities and Management Sciences (IJHMS)*., Vol. 1, Issue 1, pp 2320–4044, 2013.
[18] S. Jianga *et al.*, "An implicit control-volume finite element method and its time step strategies for injection molding simulation,“ *Computers and Chemical Engineering*., vol. 31, pp. 1407-1418, 2016.
[19] M. K. Koya, K. S. Prasad, M. S. Krishna, "DFM & SIMULATION for injection mould of knob,“ *International Journal of Mechanical and Production Engineering Research and Development (IJMPERD)*., vol. 9, Issue 5, pp. 265–276, 2019.
[20] F. Hentati, I. Hadriche, N. Masmoudi, C. Bradai, "Optimization of the injection molding process for the PC/ABS parts by integrating Taguchi approach and CAE simulation,“ *The International Journal of Advanced Manufacturing Technology.*, ISSN 0268-3768, 2019, doi.org/10.1007/s00170-019-04283-z.
[21] J.P. Beaumont, "Runner and Gating Design Handbook,“.*Hanser Gardner Publications.*, 2004:
[22] W. Dai, P. LIU, X. WANG, "An improved pin gate and its flow pattern in the cavity,“ *Journal of Injection Molding Technology.*, vol.2, Issue 2, p. 115, 2002.
[23] D. A. R.. Calhoun, J. Golmanavich, "Injection Molding Processing And Troubleshooting,“ 2002.
[24] J. Q. Li *et al.*, "Single gate optimization for plastic injection mold,“ *Journal of Zhejiang University-Science A.*, pp. 1077-1083, 2007.
[25] C. Y. Shen, "Gate location optimization in injection molding by using modified hill-climbing algorithm,“ *Polymer-Plastics Technology and Engineering.*, pp. 649-659, 2004.
[26] T.S. Li, SH. Chen, HL. Chen, "Thermal-flow techniques for sub-35 nm contact-hole fabrication using Taguchi method in electronbeam lithography,“ *Microelectron Eng*., 86, pp. 2170–2175, 2019,doi.org/10.1016/j.mee.2009.03.049.
[27] A. Ananthanarayanan *et al.*, "Mechanical Characterization of Cold Weld-lines and Meld Lines in Mesoscopic Revolute Joints for Bio inspired Structures,“ *in Proceedings of the XIth International Congress and Exposition.*, Orlando, Florida USA, 2008.
[28] SWorks, Solid works plastic (plastic databank), 2013.
[29] A. Östergen, "Prediction of residual stresses in injection moulded parts,“ pp. 1–66, 2013.
[30] T. Erzurumlu, B. Ozcelik B, "Minimization of warpage and sink index in injection molded thermoplastic parts using Taguchi optimization method,“ *Mater Des.*, 27:pp. 853–861,2006, doi.org/10.1016/j.matdes.2005.03.017