

2nd International Materials, Industrial, and Manufacturing Engineering Conference, MIMEC2015,
4-6 February 2015, Bali Indonesia

Improved gate system for scrap reduction in injection molding processes

Mehdi Moayyedien^{a,*}, Kazem Abhary^a, Romeo Marian^a

^aUniversity of South Australia, School of Engineering, Mawson lakes Campus

Abstract

This paper introduces a new geometry of the gate system in injection moulding process. The corners of existing edge gates results in the turbulence of molten plastic into the cavity which leads to internal and external defects. The aim of the new geometry is to reduce the internal and external defects of injected parts. Also easier de-gating of final part for the new geometry which decreases the visible blemish of final part after de-gating is a target of this study. A new design of edge gate is suggested for two circular flat plates with a thickness of 1mm. Internal and external defects in the plastic part are analysed with SolidWorks Plastics to validate the proposed geometry. The contribution of this research is to modify the geometry of existing edge gate by removing the corners of rectangular edge gate to reduce the scrap rate of injected parts. Also, a smooth flow of molten plastic into the cavities reduces the internal and external defects. The outcome demonstrates filling the cavities with no short shot. Also, no weld lines, meld lines or sink marks were detected with new design of the edge gate. Finally, the injected part with new design of the edge gate has less visible blemish in comparison with existing edge gate.

© 2015 The Authors. Published by Elsevier B.V. This is an open access article under the CC BY-NC-ND license (<http://creativecommons.org/licenses/by-nc-nd/4.0/>).

Selection and Peer-review under responsibility of the Scientific Committee of MIMEC2015

Keywords: Injection molding ; gate design; internal and external defects

1. Introduction

During the past century, plastics became ubiquitous into all markets. According to world consumption of raw materials by weight, plastic is the highest in comparison with other old materials such as aluminum, steel, rubber, copper, and zinc. This has resulted from its specific properties and lower production cost [1, 2]. One of the most important processes in manufacturing of plastic products is injection molding and approximately one-third of all

* Mehdi Moayyedien. Tel.: +61883025561

E-mail address: Mehdi.moayyedien@unisa.edu.au

plastics are converted into parts using injection molding [3]. Injection molding processes are growing significantly in many industries like packaging, aerospace and aviation, building and construction, automotive parts and household articles [1, 4]. The quality of the injection molding depends on the material characteristics, mold design and the process conditions [5]. The fundamental operations in injection molding are: the conversion of plastic granules into a melt, the injection of molten plastic into the mold cavity or cavities under pressure via sprue, runner and gate systems and finally the ejection of the part out of cavities by opening the mold tools [1, 6].

One of the factors which will determine the final quality of the part is the geometry and size of the gate. The main purposes of the gate are (a) to control the flow of molten plastic when the melt is forced through the compression so the temperature and shear rate will increase which results in the reduction of viscosity (b) to have a control over the melt flow both in filling and packing stages (c) to freeze the polymer at the end of injection which enables the withdrawal of the screw. In addition, the gate has significant effects on the performance of final product. The gate directly controls the flow processes in the cavity which determine the mechanical properties, dimensional stability, and the appearance of the injected part [6]. Different aspects will be evaluated for gate systems such as (a) scrap reduction; (b) easier ejection from mold tools and de-gating mechanisms; (c) central flow stream of gate with runner system to reduce the internal and external defects; (d) optimum geometry of the gate for cycle time reduction [7].

After the consideration of different geometrical schemes for gate system, a new design of the edge gate is evaluated for its effectiveness and will be discussed in the following section. The contribution of this paper is to define the new design in the gate system of injection molding. The benefits are: easier de-gating from the inject part, better attachment with part geometry like wall thickness and part specifications like appearance, reduction of the internal and external defects such as weld line, meld line, and sink marks and finally reduction in scrap. In this paper, the structure of new design of the proposed edge gate is introduced. Second the design of feeding system and injected part for two different gates is conducted using SolidWorks. To achieve an accurate result of the injection process, Finite Element Method (FEM) is employed using SolidWorks Plastic.

2. Cross sectional shape of new design of edge gate

The gate or point of initial entry of polymer melt into the cavities is a significant zone for a designer to evaluate. It should be as small as possible to avoid excess material removal and visible blemishes on the part [6, 8]. The most basic type of the gate is an edge gate which has the rectangular cross section whose main advantage is for use in multi cavity molds [9]. Generally the cross section of the gate is very important to de-gate from the injected part with minimum visible blemishes, have better attachment with part geometry, and also have a smooth entrance of molten plastic into the cavities. The proposed design of edge gate which removes the sharp corners will cover most of the mentioned characteristics of an acceptable gate.

3. Simulations

The new design of the edge gate has an elliptical cross section as shown in Fig. 1a. The mechanism of filling the mold with new design of an edge gate is different from that of the existing edge gate, resulting from a smooth entrance of molten plastic into the cavities by removing the corners of the existing edge gate. Also, de-gating of the proposed edge gate is easier than that of the existing edge gate, resulting from the proposed design of the edge gate. After designing two circular parts as two samples for this application, the next step is to simulate the injection of the part using SolidWorks Plastic. For the simulation, it is necessary to define the injection system. Hence, designing the sprue, runner and gate system should be applied as shown in Fig. 1b and 1c. The runner and gate length in total is 280mm for two circular parts with diameter of 100mm. Also, the sprue has 60 mm length with draft angle 1.5°. The length of major axis and minor axis for modified edge gate are 2.06 mm and 0.7 mm respectively.

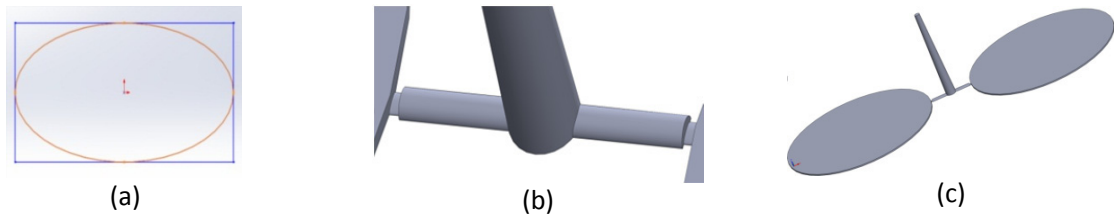


Fig. 1 (a) new design of edge gate; (b) and (c) samples of injection with sprue, runner and gate system

In order to ensure that the analysis results are accurate, FEM will play a significant role in the simulation. According to the geometry of samples, the triangle shape of surface meshes for finite element analysis is chosen as shown in Fig. 2a. The selected material for this simulation is polypropylene (P.P). Different sizes were considered for the surface mesh and finally the surface mesh size of 1mm was applied for the injected part. For the injection system which consists of sprue, runner and gate, smaller sizes were selected. This has resulted from the sensitivity of injection system as a critical area of this simulation. Hence, surface mesh size of 0.3 mm for sprue and runner and 0.2 mm for gate were selected for both elliptical and rectangular cross sectional shape of the gates.

The next stage is to set up appropriate process parameters. According to the selected materials and injection machine for this simulation, filling time is 0.55 sec, melt temperature is 230°C, mold temperature is 50°C, pressure holding time is 1.97 sec, and pure cooling time is 3.56 sec. The geometry and size of the injection system which includes sprue, runner and gate, will affect the operation cycle time, cooling time, internal defects such as sink marks, short shot etc. After running the simulation, the main factors to evaluate if the proposed edge gate is an acceptable improvement in terms of new geometry and size, are ease of fill, filling time analysis, weld and meld lines analysis, sink marks analysis, and injection pressure at the end of filling. As shown in Fig. 2b, ease of fill for an improved edge gate is at the green area which is in the most satisfactory level.

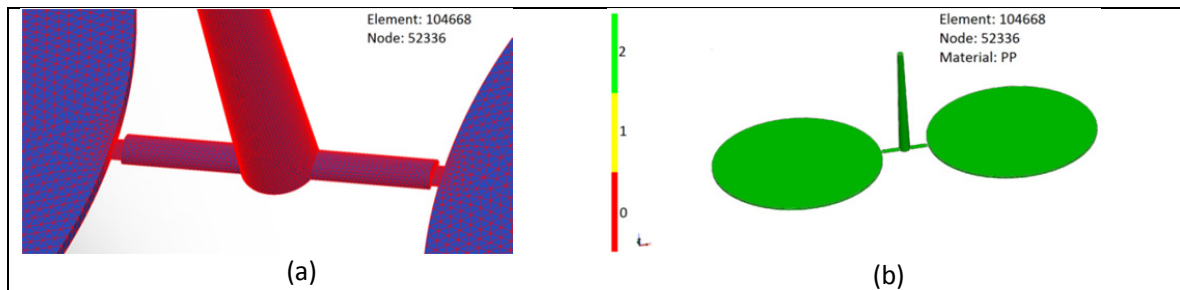


Fig. 2 (a) Finite element analysis of designed part; (b) Easy filling of injected part with an improved edge gate

To make sure that the proposed modification of the edge gate is acceptable, filling time analysis is necessary. Based on the simulation via SolidWorks Plastics, there is no difference between the filling time of the existing edge gate and proposed edge gate. Hence, the proposed edge gate does not impede its use in mass production as shown in Fig. 3. The filling time of proposed edge gate for bigger injected part will decrease in comparison with that of existing edge gate. It has resulted from reduction in the volume of proposed edge gate in comparison with existing edge gate.

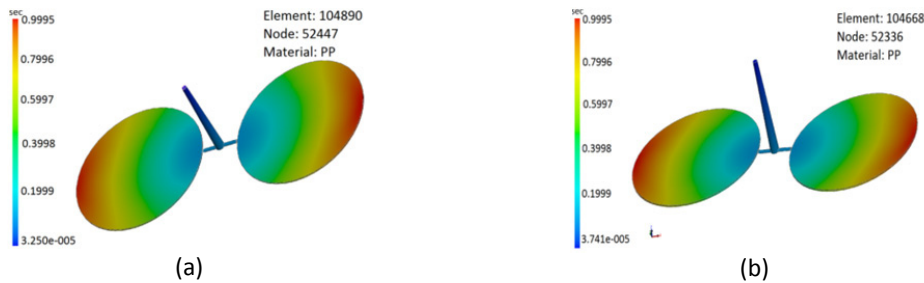


Fig. 3 filling time for (a) an edge gate and (b) an improved edge gate

The final qualities of the injected parts play a major role in plastic industry. A common external defect related to gate and runner dimension and geometry is the sink mark. A sink mark is a depression on the surface of injection molded plastic which occur during the cooling phase [10]. As shown in Fig. 4, there is a level of sink mark on the side surface of the injected part caused by the existing edge gate but with the modified edge gate, the sink mark are removed from the side surface of final part.



Fig. 4 (a) Sink mark simulation for (a) existing edge gate and (b) proposed edge gate

Weld lines and meld lines are two common defects in the plastic industry which leads to weakening or breaking of the injected parts. To differentiate the weld line and meld line, the meeting angle (θ) of the two counters of molten plastic is critically significant. If θ is less than 135° , it is considered to be a weld line but if θ is greater than 135° , it is considered to be a meld line [11, 12]. The weld line analyses of existing edge gate in Fig. 5a shows the weld lines and meld lines at the conjunction point of gate and the injected part and also weld lines at the end point of the injected parts. In contrast, there is no weld lines or meld lines for the injected part with new design of the edge gate as shown in Fig. 5b. It has resulted from lower turbulence of molten plastic into the cavities by removing the sharp corners of the existing edge gate. Also, another area which has weld lines and meld lines for both gates is the conjunction point of sprue and runner system which is not significant. It has resulted from the consideration of sprue and runner as scrap.

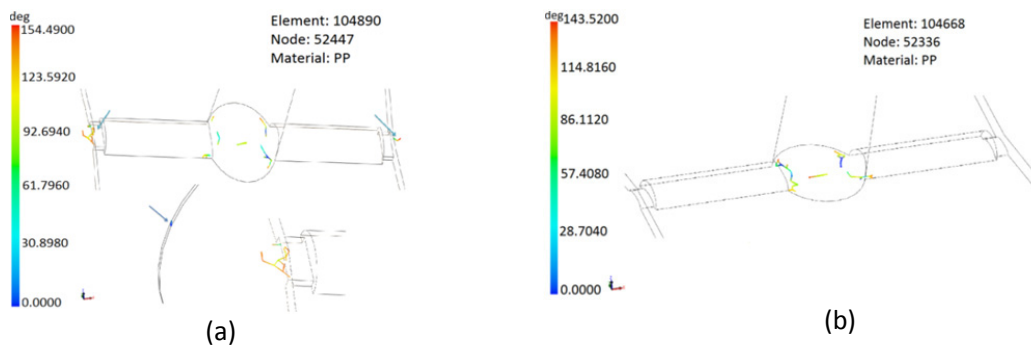


Fig. 5 weld line and meld line simulation of (a) an edge gate and (b) modified edge gate

One of the most significant factors which is affected by the size and geometry of the gate system is maximum inlet pressure. According to the simulation, the injected part with proposed edge gate can be successfully filled with injection pressure 46.3 MPa. The injection pressure is less than the max injection pressure limit which is satisfactory as shown in Fig. 6. There is a possibility to further reduce the part thickness if required. The injection pressure for existing edge gate is 44.85 MPa.

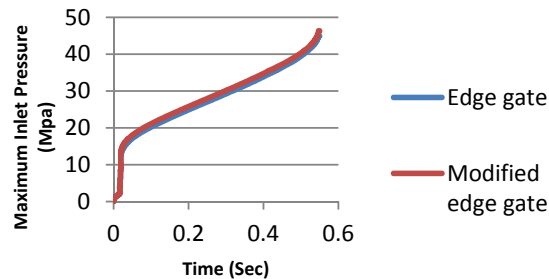


Fig. 6 injection pressure limit for edge and improved edge gate

4. Conclusion

One of the significant causes of internal and external defects in injection molding process is the size and geometry of runner and gate system. The gate system is the first orifice in a mold which causes the injection of molten plastic into the cavities. A gate system has different designs for different applications. A new design of an edge gate and existing edge gate were successfully developed via simulation with SolidWorks Plastic for two circular plates with thickness of 1mm.

The result demonstrates that the new design of the edge gate in comparison with existing edge gate has less turbulences of molten plastic flowing into the cavities, which leads to the reduction of internal or external defects in final injected parts such as weld lines, meld lines and sink marks. Also easy de-gating of the proposed edge gate has less visible blemishes on the part which leads to reduction in scrap. It has resulted from less contact surface of the proposed edge gate with the injected part. The experimental set up will be conducted as future work to validate the simulation result.

5. References

1. Zhou, H., Computer Modelling For Injection Molding. 2013: A John Wiley & Sons, Inc., Publication.
2. Salimi, A., et al., prediction of flow length in injection molding for engineering plastics by fuzzy logic under different processing conditions. 2013.
3. Tang, S.H., et al., Design and thermal analysis of plastic injection mould. Journal of Materials Processing Technology, 2006: p. 259-267.
4. Subramanian, N.R., L. Tingyu, and Y.A. Seng, Optimizing warpage analysis for an optical housing. Mechatronics, 2005: p. 111-127.
5. Altan, M., Reducing shrinkage in Injection Molding via the Taguchi, ANOVA and neural network methods. 2010: p. 599-604.
6. DAI, W., P. LIU, and X. WANG, An improved pin gate and its flow pattern in the cavity. Journal of Injection Molding Technology, 2002.
7. Jones, P., Mould Design Guide. 2008.
8. Calhoun, D.A.R. and J. Golmanavich, Injection Molding Processing And Troubleshooting. 2002: Ron Jon.
9. Beaumont, J.P., Runner and Gating Design Handbook. 2004: Hanser Gardner Publications.
10. Goodship, V., Troubleshooting Injection Moulding. 2004.
11. Ananthanarayanan, A., 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 2008. Orlando, Florida USA.
12. works, s., solid works plastic (plastic databank). 2013.