RESEARCH PROGRESS IN MECHANICAL AND MANUFACTURING ENGINEERING VOL. 2 NO. 1 (2021) 193-202

RPMME

Homepage: http://penerbit.uthm.edu.my/periodicals/index.php/rpmme
e-ISSN : 2773-4765

The Comparison on Mold Design for Injected Molded Integral Hinges Test Sample Via Autodesk Mold Flow Adviser 2019

M. Nidzuwan[1], **M. H. Othman**[1*]

[1]Faculty of Mechanical and Manufacturing Engineering,
University Tun Hussein Onn Malaysia, 86400 Parit Raja, Johor, MALAYSIA.

*Corresponding Author Designation

DOI: https://doi.org/10.30880/rpmme.2021.02.01.022
Received 01 March 2021; Accepted 01 April 2021; Available online 15 April 2021

Abstract: Plastic injection molding has been in high demand today. In this study, the Moldflow Adviser is used to make the comparison on mold design for injected molded integral hinges test sample. The injection molding machine and product used in this research are Nissei NP7F and integral hinges test sample that available in UTHM lab. Computer models or simulation in injection molding can be categorized into three groups, such as, modeling, optimization and process control. The designs that are used of injection molding are divided into six design which involves change of feeding system parameter. On this study, processing conditions selected were filling time, filled temperature, weld lines, warpage, air traps, volumetric shrinkage and product quality. Before start the mold design process condition in simulation software, integral hinges test sample has been drawn in SolidWorks software first then transfer to Moldflow Adviser software in STL file format. For this study, feeding system has been change for every design. In order to obtain the data of product, the evaluation of criteria analysis have been made. As the result, comparison of product quality analyzed. In a conclusion, the result of comparison indicated that Design 6 have the high product quality with the uniform filled temperature 222.7°C, lowest volumetric shrinkage 12.14%, and 48 air traps only. However, Design 2 has advantage on lowest warpage values and Design 3 are good in filling time and lowest weld lines values compared to Design 6.

Keywords: Mold Design, Integral Hinges, Polypropylene, Shrinkage, Warpage, Gate Location, Air Traps, Feeding System.

1. Introduction

Injection molding is one of the common methods of mass production of plastic products. The same element or parts has been produced thousands or even millions of times in succession. The decisions taken in the designing phase cost relatively least in terms of the total expense of the project, which have a significant influence on the quality of the products [7]. It is mean, the price of a product through the

*Corresponding author: hilmi@uthm.edu.my

injection molding process will be lower once the design phase is reached. But designing plastic components is a dynamic activity containing several variables that tackle the collection of product specifications. In addition to practical and structural problems, manufacturing problems play a major role in the design of the injection molded plastic component. To solve this problem, simulations have been used in this design phase. The use of simulation can reduce the cost of mold design in designing phase.

Computer simulation for injection molding has been an important field of study for several years. In my view, computer models in injection molding can usually be categorized into three groups, for example, modeling, optimization and process control [5]. In particular, the advantages of computer modeling are accentuated by the fact that, in order to produce a single part for evaluating its performance, first a custom mold must be constructed, which can cost tens to hundreds of thousands of dollars. This is typically several million times the selling price of the product to be produced. When the component does not fulfill requirements, it can be quickly adjusted or the second element can be machined to represent a different specification. Thus, if the first parts do not work, the investment in engineering and machine time is minimal compared to the construction of the mold [1].

2. Methodology

The methodology of this study is design to meet the scope of the study, which is also, is to evaluate the step that will be conduct during this study will be explain in this section. Development of mod design is the main step in this study.

2.1 Development of Mold design

SolidWorks was used to design geometric structure of the integral hinges test sample as well as the MoldFlow application was used for injection analysis of simulation molding.

2.1.1 Type of analysis

In MoldFlow Adviser an advanced option like ‘Automatic clean up’ to fix imported model defects and ‘Check Suitability’ to determine whether it is suitable 3D or Dual-Domain analysis need to be selected for getting more accurate results. According to Moldflow system, it recognized that this integral hinge test sample is suitable for Dual Domain analysis because it has a thin wall model.

2.1.2 Materials selection

In analyzing this integral hinge test sample, the material used in this research are Polypropylene (PP) or also stated as Generic PP in Moldflow Adviser software. Table 1 shows the mechanical properties of Generic Propylene used in this project.

Table 1: Mechanical Properties of Generic Polypropylene

No	Properties	Value
1.	Elastic modulus	1340 MPa
2.	Poisson ratio	0.392
3.	Shear modulus	481.3 MPa
4.	Mold surface temp. (recommended)	20 – 60 ℃
5.	Melt temperature (recommended)	220 – 240 ℃
6.	Ejection temperature	101 ℃

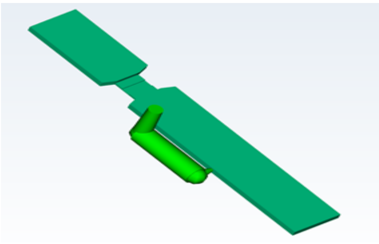
2.1.3 Feeding system

The importance part in mold design are gate, runner and sprue that can be categories as feeding system. Gate and runner are closely related that bring the molten materials. Thin or small passage of gate bring molten material via runner to the mold cavity. Table 2 shows the different of mold design specification based on feeding system.

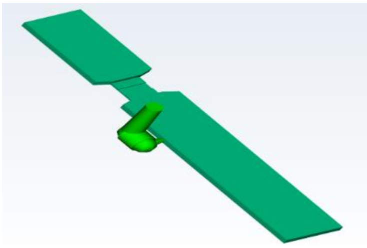
Table 2: Specification of integral hinges test sample mold design.

Specification	Design 1	Design 2	Design 3	Design 4	Design 5	Design 6
Length of Runner	16 mm	4 mm	26.5 mm	16 mm	4 mm	26.5 mm
Runner Diameter		5 mm			4 mm	
Height of Sprue	12.5 mm	12.5 mm	12.5 mm	12.5 mm	12.5 mm	12.5 mm
Length of Gate	4 mm	4 mm	4 mm	4 mm	4 mm	4 mm
Material	Generic PP					

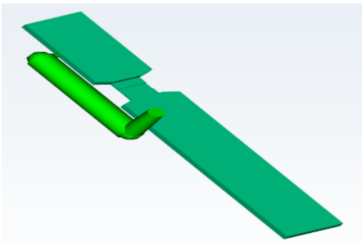
There are 6 design of mold design that were simulated. The 6 design of the mold design was simulated with each one of them have different parameter of feeding system. Figure 1 shows every type of mold design.



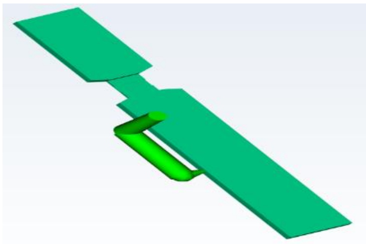
Design 1



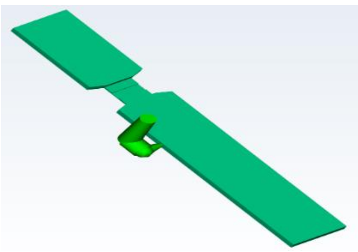
Design 2



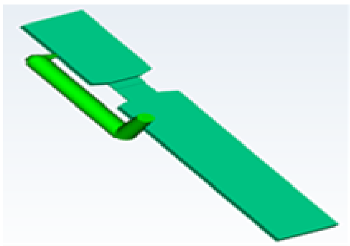
Design 3



Design 4



Design 5



Design 6

Figure 1: Type of mold design

3. Results and Discussion

3.1 Simulation result

Flow analysis or simulation is a computational simulation that offers a way to simulate how plastic can flow through the injection mold and how this process can affect everything from filling pressure to shrinking, cooling, and eventually warping. Table 3 shows the result of simulation finding using Moldflow Adviser 2019 for six design that involved. The parameters represent filling time, temperature when filled, volumetric shrinkage, warpage and weld lines.

Table 3: Result of Simulation

Parameter Quality or defects	Design 1	Design 2	Design 3	Design 4	Design 5	Design 6
Filling time (s)	0.605	0.415	0.660	0.582	0.409	0.615
Temperature when filled (°C)	209.5 - 222.3	193.9 - 225.4	222.6	209.4 - 222.4	193.8 - 225.4	222.7
Volumetric Shrinkage (%)	14.12	13.29	12.15	14.09	13.32	12.14
Max. Warpage (mm)	0.616	0.577	0.696	0.616	0.578	0.699
Weld lines (deg)	1.318 - 34.74	1.410 - 34.81	1.113 - 34.58	1.212 - 34.66	1.413 - 34.81	1.144 - 34.61

The filling time result indicates the direction of the flow front at frequent intervals as the cavity fills. Figure 2 shows the different of filling time for every design.

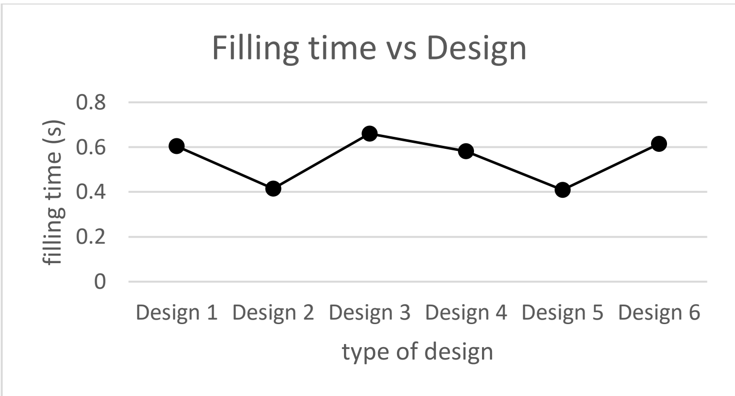


Figure 2: Graph of Filling time(s) vs Designs

The higher time of the injection, the higher the strength of the product can be producing. The longer time of the injection affected the lowest defects and the optimum performance of the mechanical properties [2]. The result show, design 2 has a very shortest filling time and Design 3 has the longest filling time. If the part is a short shot, the segment that has not been filled has no color. Fill time is the time needed to fill the portion inside the cavity, to demonstrate how the plastics melt flows to fill the cavity.

Volumetric shrinkage's evaluation is very important in determining the quality of a plastic products. This is because excessive wall thickness and insufficient packing can both contribute to high volumetric shrinkage of the part [3]. Based on the Figure 3, if evaluated in terms of percentage, can see that Design 6 has the advantage of the lowest volumetric shrinkage percentage of 12.14% only, followed by Design 3. while Design 1 occupies the highest percentage of 14.12% volumetric shrinkage.

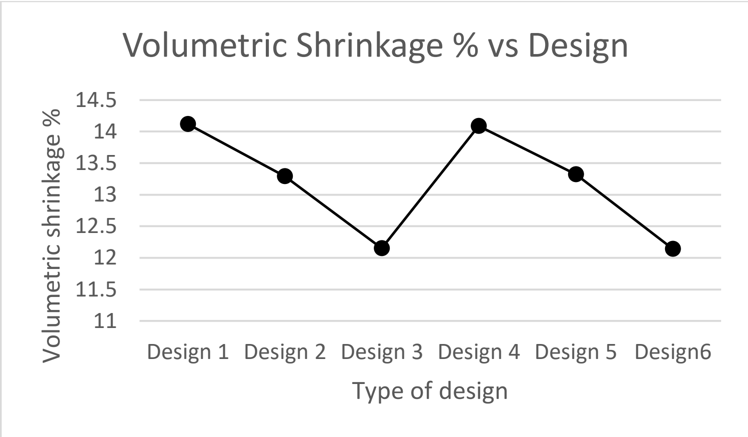


Figure 3: Graph of Volumetric Shrinkage (%)

According to Figure 4 the resulting graph, it can be seen more clearly that designs 3 and 6 have the highest warpage rates compared to designs 2 and 5 which have the lowest warpage rates. Warpage is one of the adverse effects of shrinkage problems that occur on the product. Areas where high warpage occurs are areas where there is a lot of shrinkage happens because there are differences in internal tension in the material due to difference in shrinkage [6].

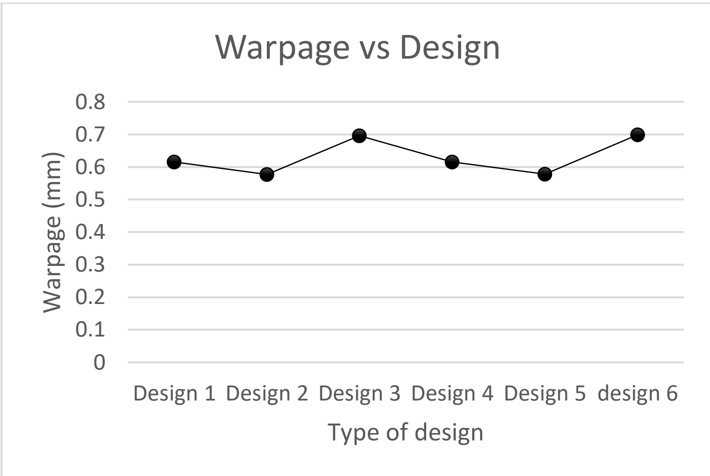


Figure 4: Graph of Warpage (mm)

A weld line on plastic parts may cause structural problems or is simply unacceptable. The welding line happens when two flow fronts cross, and the polymer molecules are misaligned. The sharp difference in molecular orientation at the weld induces a major decrease in strength at this stage.

Based on Figure 5, it appears that the appearance of weld lines on the integral hinges varies according to the position of the gate and diameter of runner on the integral hinges. Design 3 and 6 show less value and number of weld lines than other designs. To eliminate or reduced the weld lines on integral hinges, maybe need to increase the diameter of the gates and the runners to make it easier to pack the part.

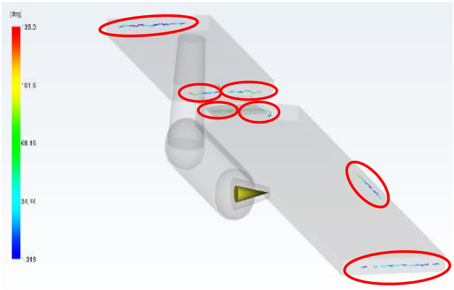
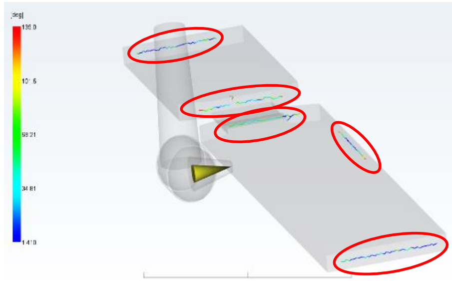
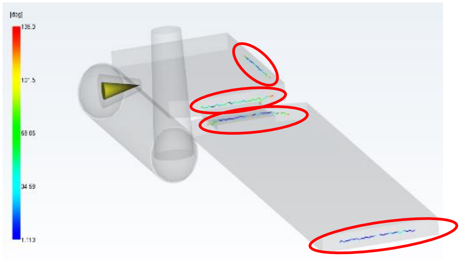
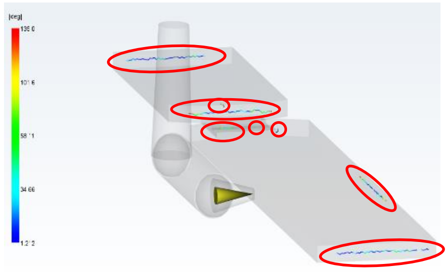
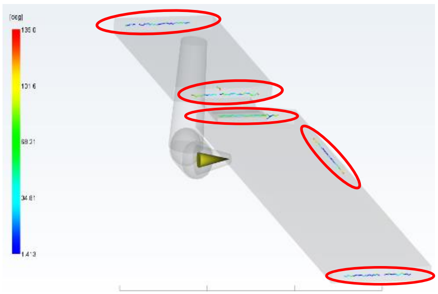
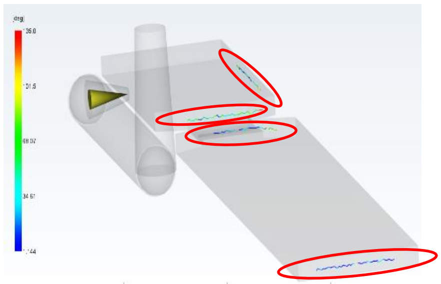
Design 1 	Design 2 
Design 3 	Design 4 
Design 5 	Design 6 

Figure 5: Formation of Weld Lines

Figure 6 shows the total of air traps in integral hinges test sample for every type of mold design. The trapped air cause incomplete filling and packaging, and also cause surface bloating in the final component. From the graph in figure, Design 6 have shown the least amount of air trap compared to the other design with only 48 air traps. The air trap at the critical position is also only 22 at the middle and hinges of the product. However, Design 4 shown the most air trap with 68 air traps. With only 48 air traps in product will make Design 6 will be among the best to choose because air traps will cause a short shot (incomplete fill), and surface defects such as burn marks [4].

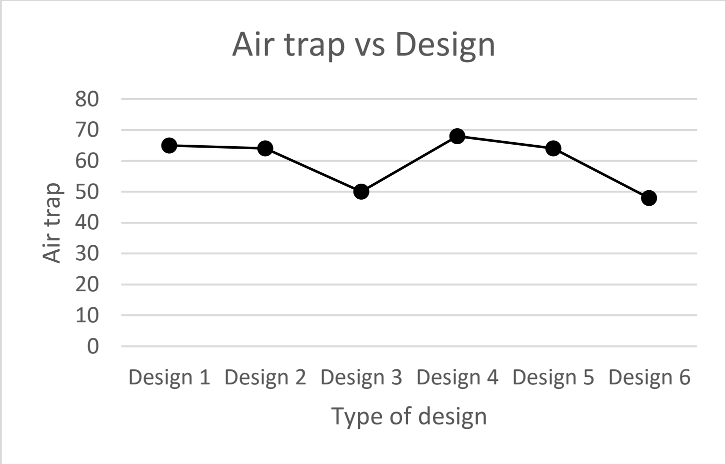


Figure 6: Graph of Total Air Trap

In determining the level of quality of a product, quality prediction has been used. The results of the quality prediction are used to estimate the quality of the mechanical properties and appearance of the component or product. Figure 7 shows the prediction of integral hinges test sample quality.

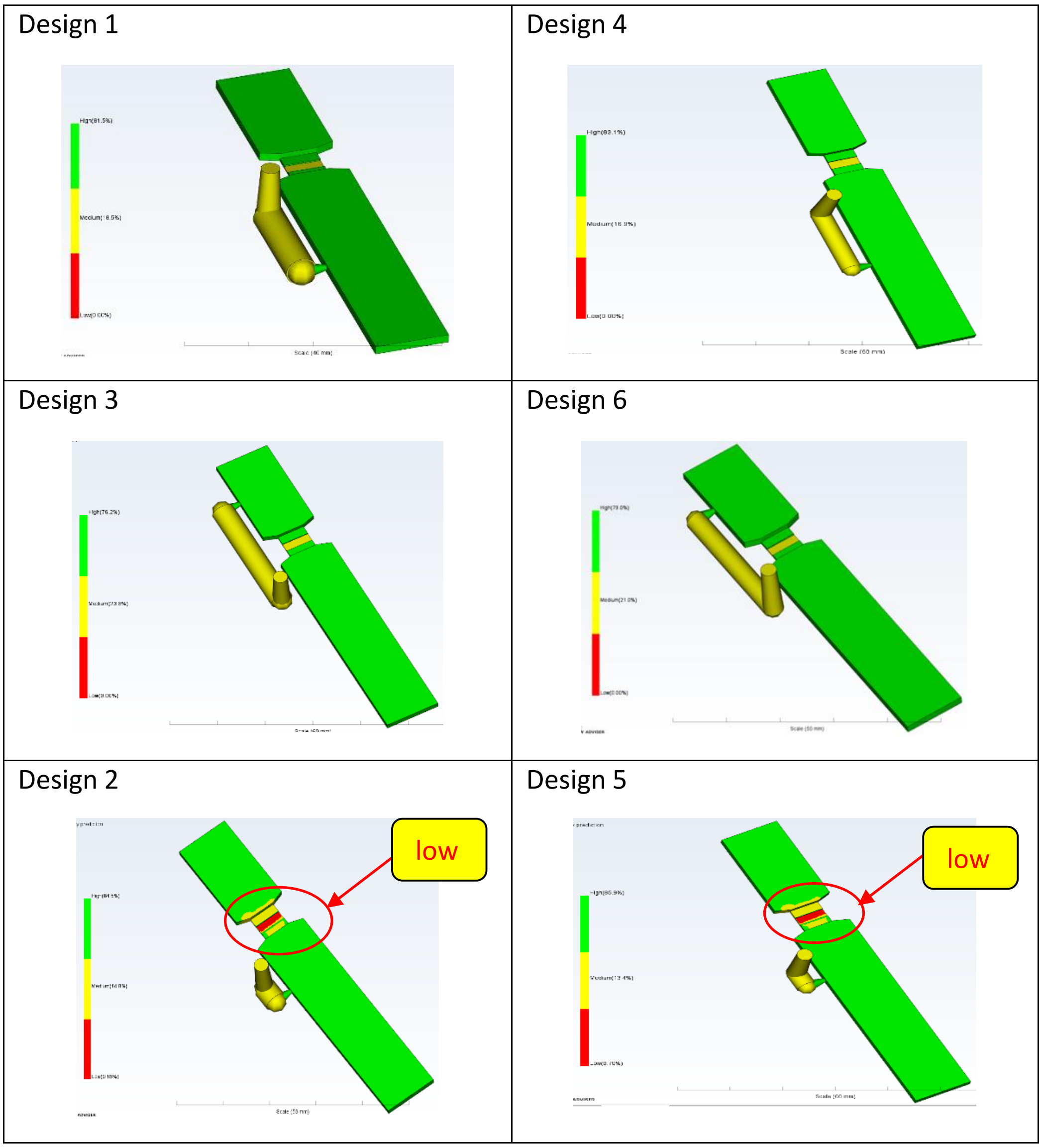


Figure 7: Quality Prediction of Integral Hinges Test Sample.

From the Figure 7, the product with all green can be acceptable. Some yellow on the product also can be acceptable but as the percentage of yellow increases, the part quality will decrease then it will not be acceptable. However, if the product has some yellow and red, the standard of the product is more likely to be unacceptable. If the ratios of yellow and red rise, the quality of the component will decrease. Attention should be paid to the feed system, as low design quality will have an impact on the component and decrease quality. Design 1,3,4 and 6 will be acceptable based on the results

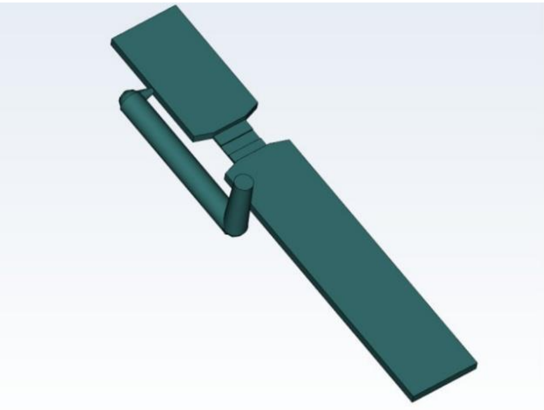
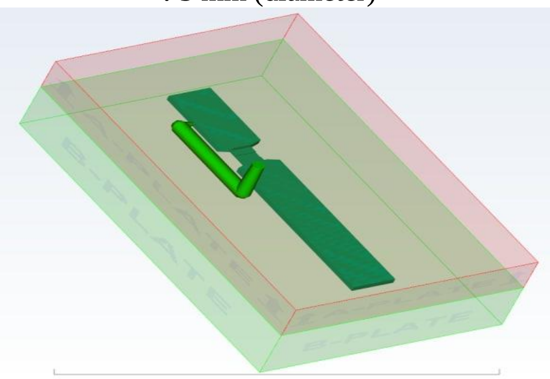
In making the comparison or choosing a mold design, various criteria should be emphasized as found in this study. This process commonly used in design decision making and the best design for the mold design can be determined. The parameters involved are filling time, filled temperature, volumetric shrinkage, warpage, weld lines, air trap and product quality. Table 4 shows the design selection process according to the simulation findings.

Table 4: Design Selection based on Simulation Results.

Designs **Parameter**	**Design 1**	**Design 2**	**Design 3**	**Design 4**	**Design 5**	**Design 6**
Filling time (s)	0.605	0.415	0.660	0.582	0.409	0.615
Temperature when filled (°C)	209.5 - 222.3	193.9 - 225.4	222.6	209.4 - 222.4	193.8 - 225.4	222.7
Volumetric Shrinkage (%)	14.12	13.29	12.15	14.09	13.32	12.14
Max. Warpage (mm)	0.616	0.577	0.696	0.616	0.578	0.699
Weld lines (deg)	1.318 - 34.74	1.410 - 34.81	1.113 - 34.58	1.212 - 34.66	1.413 - 34.81	1.144 - 34.61
Air traps	65	64	50	68	64	48
Quality	High	Low	High	High	Low	High

Based on Table 4, Design 3 has the best filling time parameter because of the longest filling time 0.660 s than others. For the parameter of filled temperature, temperature 222.7°C in design 6 is the best because it almost achieves high temperature stability on the whole product compared to other designs. Design 6 also has the lowest Volumetric shrinkage value of 12.14%. However, the value of warpage in Designs 3 and 6 is higher when compared to design 2 which has the lowest value of warpage. In terms of the value of weld lines formed, Design 3 is in the best position by showing the lowest average value. If evaluated on others parameters like the number of air traps formed, Design 6 has been selected because it only has 48 air traps on the entire product followed by a high product quality rate. As the result, the best mold design among others is the mold design 6 that has the best parameter value. Table 5 show the final selection of mold design for integral hinges test sample in this analysis.

Table 5: The Selected Mold Design

Result	Description of mold design
 **Selected design:** Design 6 **Runner type:** Cold (Circular shape) **:** 5 mm (diameter)  **Mold design:** 100 x 70 x 25 mm **Melt temperature:** 220 °C **Mold temperature:** 50 °C	**Simulation result** **Filling time:** 0.615 s **Filled temperature:** 222.7 °C **Volume Shrinkage:** 12.14% **Maximum Warpage:** 0.6986 mm **Weld line:** 1.144 deg~34.61 deg **Air trap:** 48 **Part Quality:** High The selection of this design is based on the previous study and the theory which includes: • The filling time is faster than Design 3 and contribute to the lower defects. • The balance of temperature when filled is good. • The less of volumetric shrinkage for the entire Integral Hinges test sample product. • The least trapped air content in the product. • This design produces less defects and high quality based on the part quality prediction. Weakness of this design during the simulation which includes: • This design is the highest warpage on the product because there are differences in internal tension in the material due to difference in shrinkage. • High warpage will make the residual stress is released from the moulding process, which induces deformation. • High weld lines will be reduced strength, as well as burn-mark and bubble defects.

4. Conclusion

In the conclusion, the investigations have been performed to allow a comparison of the mold design of the feed system based on many criteria including the gate, runner and injection location. As a result, it was observed that the parameter of the gate and the location of the injection should be correctly positioned to reduce or eliminate the formation of defects in the Integral hinge test sample. In results, Design 6 was selected from the simulation for the mold design, which has a good design consistency.

Acknowledgement

The authors would also like to thank the Faculty of Mechanical and Manufacturing Engineering, University Tun Hussein Onn Malaysia for its support.

References

[1] Beaumont J.P., Nagel R., Sherman R., Successful Injection Molding. 2002, Munich: Hanser Publishers.

[2] Bryce, DM. 1998. III Plastic Injection Molding: Mold Design and Construction Fundamentals. http://books.google.com/books?hl=en&lr=&id=uiPVsJN5J8sC&oi=fnd&pg=PR15&dq=Plastic+Injection+Molding+...+mold+design+and+construction+fundamentals&ots=JkrN2O-I_h&sig=ORs8RQvKxeP3yijigrXx_YWlqpg.

[3] Chang TC (2001) Shrinkage behavior and optimization of injection molded parts studied by the Taguchi method. PolymEng Sci 41(5):703–710

[4] Fu, M W, A Y C Nee, and J Y H Fuh. 2002. "The Application of Surface Visibility and Moldability to Parting Line Generation." 34: 469–80

[5] Huamin Zhou (2013). Computer Modeling For Injection Molding. A John Wiley & Sons, inc., Publication.

[6] Liao SJ, Chang DY, Chen HJ, Tsou LS, Ho JR, Yau HT, HsiehWH (2004) Optimization process conditions of shrinkage andwarpage of thin-wall parts. Polym Eng Sci 44(5):917–92

[7] Yoder, Paul R., David M. Stubbs, Kevin A. Sawyer, and David Aikens. 2017. 1 Fourth Edition: Opto-Mechanical Systems Design: Design and Analysis of Opto-Mechanical Assemblies Opto-Mechanical Design Process.