Research Progress in Mechanical and Manufacturing Engineering Vol. 3 No. 1 (2022) 539-549
© Universiti Tun Hussein Onn Malaysia Publisher's Office

RPMME

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

Design A Mould Insert for A Complex Design

Mugilvarnan[1], Dr Halim Irwan Ibrahim[1]*

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

*Corresponding Author Designation

DOI: https://doi.org/10.30880/rpmme.2022.03.01.056
Received 15 Nov. 2021; Accepted 15 April 2022; Available online 30 July 2022

Abstract: Cycle time of a part in injection moulding process is very important as the rate of production and the quality of the parts produced depends on it, whereas the cycle time of a part can be reduced by reducing the cooling time which can only be achieved by the uniform temperature distribution in the moulded part which helps in quick dissipation of heat and locating a proper gate system[1]. The aim of the project is to design and conduct simulation using SolidWorks 2019 and Autodesk Moldflow Advisor software of a mould insert for car piston head material for injection moulding. The project focuses on incomplete filling and warping which caused by the gate location and cooling channel of the mould insert. Two types of gate locations were tested in this project. The purpose of designing this mould is to improve the reliability and productivity during injection moulding process by reducing the cycle time of the part in the injection moulding. This was analysed using Autodesk Moldflow Advisor and finished model was designed in SolidWorks 2019 software. The gate location was affecting the fill time, quality prediction, time to reach ejection temperature, weld line, as well as injection pressure. The cooling channel was tested for volumetric shrinkage. In Autodesk, fill + pack, cool analysis and warp were analysed in the simulation. The gate location and cooling system effects the part quality of a mould insert.

Keywords: Injection Molding, Fill Time, Volumetric Shrinkage, Warpage, Gate Location, Cooling Channel

1. Introduction

Injection moulding is one of the most abused mechanical processes in the creation of plastic parts. Its prosperity depends on the high capacity to create 3D shapes at higher rates than, for instance, blow moulding. The fundamental rule of injection shaping is that a solid polymer is liquid and infused into a cavity inside a shape, which is then cooled also, the part ejected from the machine. The basic principle in an injection shaping process includes filling, cooling and also ejection. The cost-effectiveness of the process is depending on the time consume in the moulding cycle. Correspondingly, the cooling process is the main process among the three, it decides the rate at which the parts are produced[2]. As in most current ventures, time and costs are connected unequivocally. The more is the time to produce a part, the more expensive. A decrease in the time spent on cooling the part before it is ejected would radically

*Corresponding author: Ibrahim@uthm.edu.my

build the production rate, consequently, reduce the costs. It is along these lines critical to comprehend and in this manner streamline the heat transfer processes inside a common ejection process efficiently[3].

Gates are the transition zone between the runner device and the cavities, they must ensure uniform flow filling of the cavities, and they must prevent jetting into the cavities with plastics. Gates are better linked to the runners in two keyways, as a centric gate and an eccentric gate, because of this feature. Therefore, careful selection, position and location in all kinds of gate systems until selecting the gate position, the placement of the gates on the moulded part and the runner system results in the net finishing of the mould part and optimization of the position is needed[4]

Moreover, in the cooling phase, heat transfers between the molten material inside the cavity and the cooling fluid (generally water) flowing through the cooling channels inside the mold, until ejection temperature is achieved and part is stable enough for demolding. Thus this rate of heat exchange is very important and directly related to the time taken by the cooling phase. So it is important to understand and optimize the cooling channel design to optimize the rate of heat transfer in an injection molding process. Proper design of the cooling channel is required for a faster cooling phase[5].

Thus, this study focusses on reducing the warpage of the injected parts and volumetric shrinkage by using a proper gate location and cooling system. This paper describes the analytical study of cooling analysis and locating of gate system applied to the modelled piston head using Autodesk Moldflow Advisor. The fill time and temperature variance have been studied for two different cooling channels.

2. Materials and Methods

Figure 1 shows the overview of the methodology involved in the present study. The activities start with material selection. Polycarbonate will be chosen as a plastic material which poured through the channel called sprue for this project. The step followed with design selection and specification. There will be some parameters that would be consider in this study. The parameters are gate design, gate location, runner, sprue, injection pressure and mould temperature. Determining the cavities is required several limitations, the number of the cavities and mould construction depend on both economic and technical factors, quantity of parts to be moulded at one cycle, required cycle time. Since, in this project only one part needs to be moulded, single cavity and core has been choosing.

Moreover, the hot runner system is suitable for this project. The hot runner system is required several factors to be considered. One of the major factors to be considered is cycle time. Hot runner system has reduced the cycle time and make the production faster. The internally heated hot runner system is compatible for this project because it reduces the heat loss for the entire mould. In addition, there will no material stick through the runner system or mould because Polycarbonate (PC) is thermally stable.

Lastly, sprue connects the nozzle of the injection moulding machine to the main runner or cavity. The sprue should be as short as possible to minimize material usage and cycle time. The bushing should have a smooth, tapered internal finish that has been polished in the direction of the draw to ensure clean separation of the sprue and the bushing.

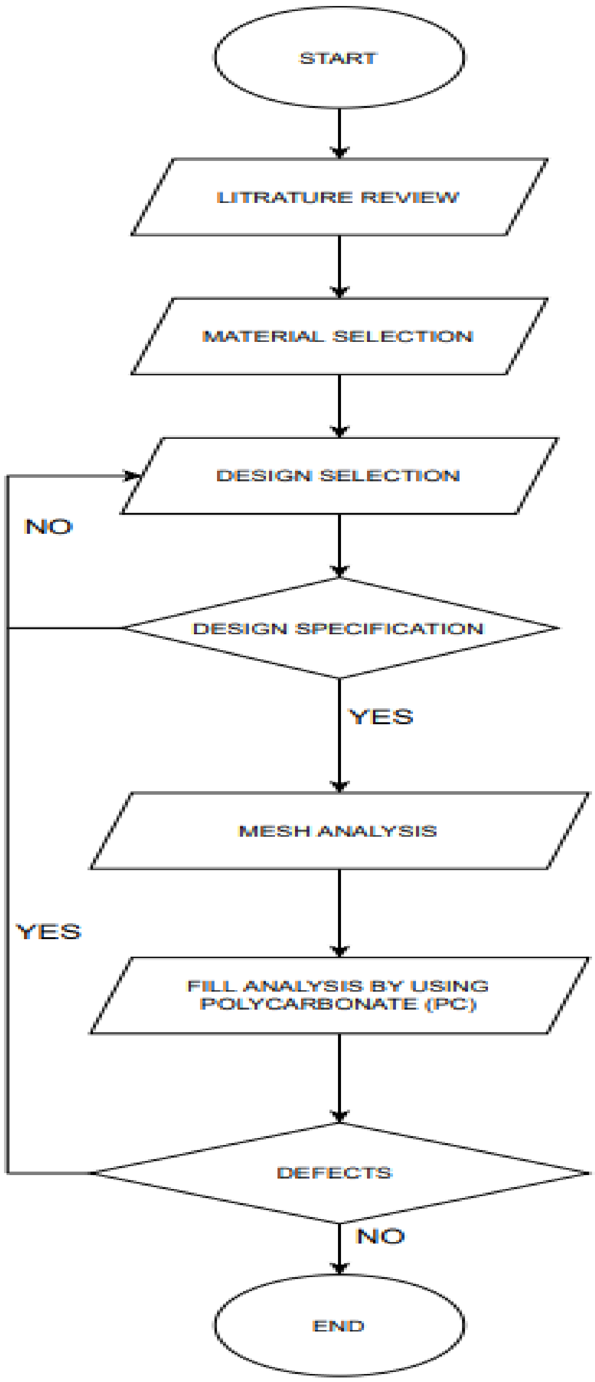


Figure 1: Methodology of the present study

2.1 Model

The mould that designed in this project is for piston head. Piston head act as a heat shield between the combustion chamber and the portions of the lower piston. The head region diameter of the piston is 80.6mm and head region height is 8mm.

Pressure is the driving force that overcomes the resistance of polymer melt pushing the polymer to fill and pack the mould cavity. Injection pressure should be low initially and increased to the point of filling the part. A very high injection pressure required in range 130 MPa to 180 MPa as the material is extremely viscous. Sometimes it depends on the wall thickness.

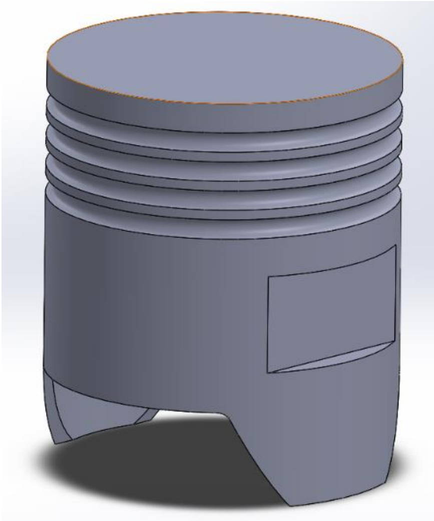


Fig. 2. Piston head

The mould temperature for this project is 90^0C due to polycarbonate. Reinforced materials require higher mould temperature than nonreinforced materials. Higher mould temperature will achieve a smoother, more blemish-free surface providing a resin rich skin on reinforced materials. Mould temperatures make sure not lower than 85^0C. Mould filling and article quality will improve with increasing temperature.

2.2 Heat Exchange System

Cooling channel for this project is having to be uniform mould cooling. The uniform mould cooling is achieved if cooling channels connects in a one loop from the coolant inlet and to its outlet. The cooling channels are in uniform in size, so the coolant can maintain its turbulent flow rate to its entire length. The temperature difference of the coolant at the inlet and outlet should be within the range of 50C for general moulds and 30C for precision mould. Two different cooling has been selected for this study which are conformal cooling system and conventional cooling channel (CCC).

The conventional cooling channel made in this study are shown below. This cooling channels had the same diameter of 10 mm. For these channels, distance between cooling channels centres were 6 mm. The coolant Reynolds number was calculated to be 1664990 and temperature at coolant inlet was 25 ° C, which indicates that flow of water was fully turbulent.

The conformal cooling channel made in this study are shown below. This cooling channels had the same diameter of 10 mm. For these channels, distance between cooling channels centres were 6 mm. The coolant Reynolds number was calculated to be 1664990 and temperature at coolant inlet was 25 ° C, which indicates that flow of water was fully turbulent.

2.3 Gate Location System

For determining the ideal gate location with the help of Autodesk Moldflow advisor, the basic part details are required. The gate location examines these five aspects of the part:-

i. Process Ability
ii. Minimum Pressure
iii. Geometric Resistance
iv. Thickness
v. Flow Resistance Areas

So, once the input details are given, the advisor suggests the ideal gate location for the product.

In this study, the best position of the gate is found by trial and error and different positions for the gate are suggested and visualize via inspection of the model part in terms of quality, air traps, weld line, time to reach ejection temperature and fill time. With the gate placed in the centre and the bottom surface of either of these two solutions are possible: internal position and external position show the best position of gate system

2.4 Simulation and analysis using Autodesk Moldfllow Adviser

To study fluid flows, flow domains are divided into smaller subdomains (made up of geometric primitives such as hexahedra and tetrahedral in 3D and quadrilaterals and triangles in 2D) which are collectively known as meshing. There are three different ways to mesh the product design in the mold flow program. These are Midplane Mesh, Dual Domain Mesh, and 3D Technology.

The mesh type used for this project is Dual Domain Mesh, this type of mesh is known to be a surface mesh along with the Midplane type. It provides the basis for the Dual Domain analysis and includes a mixture of different forms, including regions with typical Midplane elements and surface shell elements. This type of mesh is suitable when there are several thin regions in the design of the component. The reason for choosing this sort of mesh is that it does not take too much time to get an analysis.

The major benefit of doing fill analysis is predicting a fill pattern that will help us understand some of the defects, if any, in the product. Other advantages include scrap reduction, balance and pressure distribution, material selection, clamp strength determination, welding, and gas trap position detection and the determination of shear stress levels.

In this project, Autodesk Moldflow Adviser were used to do simulation. This software can stimulate the filling phase of the injection molding process depending on the assigned thermoplastic material and process parameters.

2.5 Procedure of simulation

The part geometry was created in the CAD system. The drawing in the solid work software should save in STL format because Autodesk Moldflow adviser cannot read the normal SLDPRT format. The piston shape was created using solid work software. Then, the solid model is transferred to Autodesk Moldflow software in STL format for the simulation. The figure shows process flow to run simulation.

The simulation starts with select part geometry. Then, location of the injection point are determined in 3 axis, X,Y,Z. Figure, show the example of injection point that appears on the screen of Autodesk Moldflow Adviser and can define the gate location after this. Next, we need to draw the runner and sprue then set location by previous steps. Then the process condition is defined that involves pack and warp, sink marks, cooling time, melt temperature, mold temperature, injection pressure and material going to be used. Then the simulation will start through the analyze button. The final result detail pops up after the simulation done.

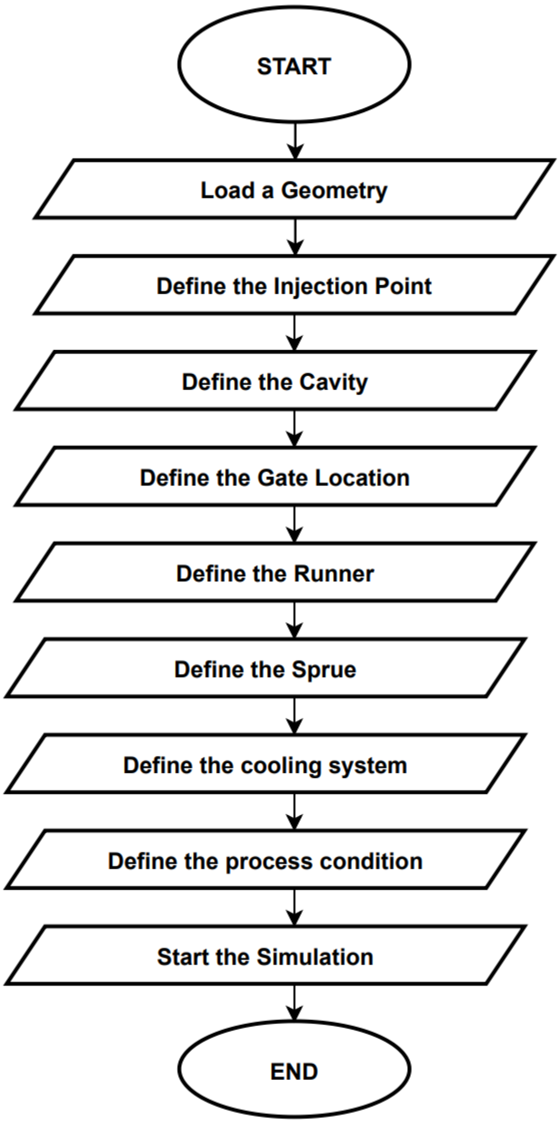


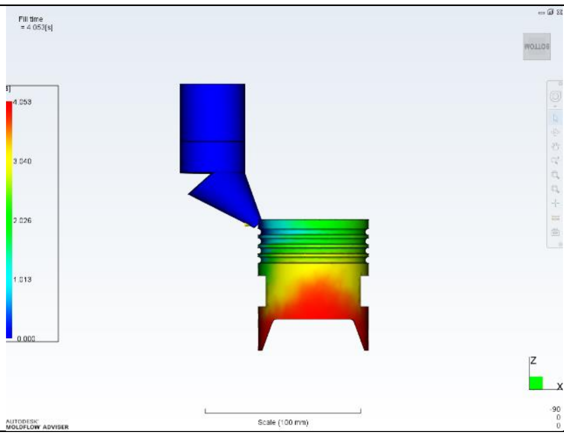
Figure 3: Flowchart of procedure of simulation

3. Results and Discussion

In this chapter, the Autodesk Moldflow Adviser software is employed on the second phase of the project for the simulation and analyzing. The simulation enables to optimize the process parameter such as filling process, filling time, temperature when filled, injection pressure, mold temperature, melt temperature and the quality of the design, whereas the cycle time of a part can be reduced by reducing the cooling time which can only be achieved by the uniform temperature distribution in the moulded part which helps in quick dissipation of heat and locating a proper gate system. The focuses on incomplete filling and warping which caused by the gate location and cooling channel of the mould insert. Two types of gate location were tested in this project. In Autodesk, fill + pack, cool analysis and warp were analysed in the simulation by the gate location and cooling system effects

3.1 Gate Design and Location

<table>
<tr>
<td>Location A:

The filling time of this design is 4.06s. This filling time is the lower than gate location B. this gate location requires less time to fill the cavity.</td>
<td>
</td>
</tr>
<tr>
<td>Location B:

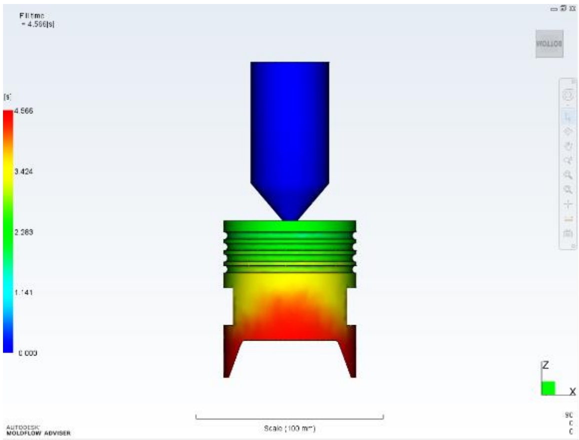
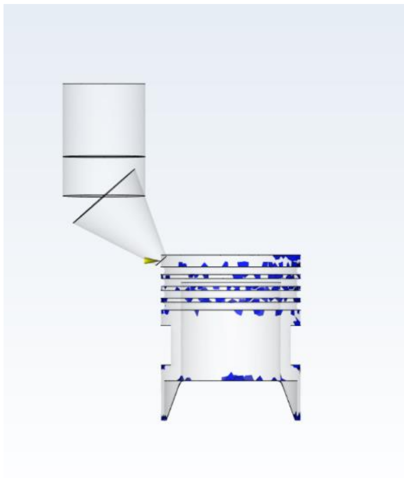
The filling time for this is 4.56s. The filling time is higher than gate location B because improper gate location.</td>
<td>
</td>
</tr>
</table>

Figure 4: Filling time of both design

<table>
<tr>
<td>Location A

There are many air traps in this gate location. However, the air traps are lower than gate location B.</td>
<td></td>
</tr>
</table>

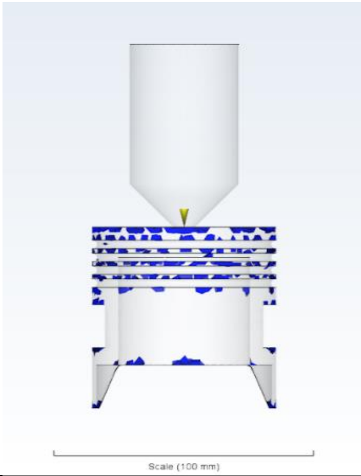
Location B There are many air traps in this gate location. The air traps are greater compared to gate location A.	

Figure 5: Air traps for both design

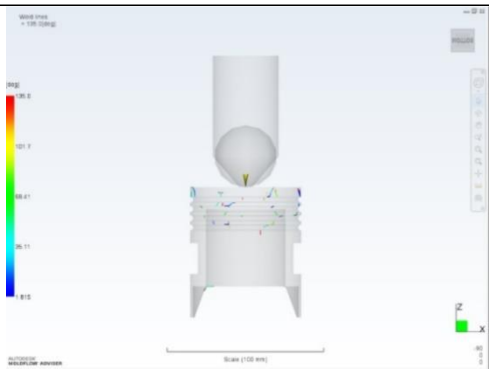
Location A The gate location has less weld line compared to gate location B. this because of the gate is location perfect for this design	
Location B The gate location B has more weld line compared to gate location B. this is because of improper gate location.	

Figure 6: Weld line for both design

3.2 Cooling System

The series cooling system was designed for this project. At the underground stage, an effective cooling system must be used to maintain this optimum temperature with uniformity and heat dissipation. Two types of cooling system were designed for this project which are conventional and conformal. The temperature variance and volumetric shrinkage at different cooling system is shown below

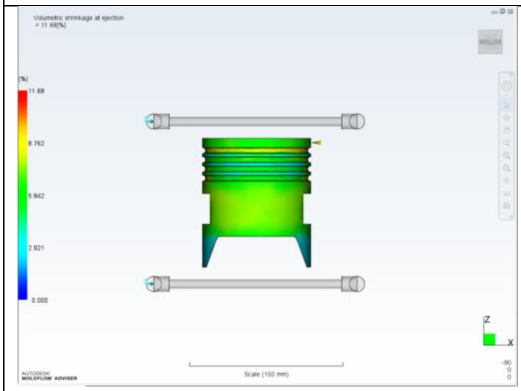
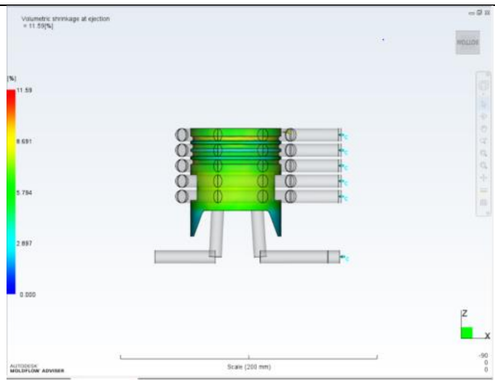
Conventional cooling system	*Conformal cooling system*
	

Figure 7: Volumetric shrinkage at ejection

When using the conventional cooling system, the volumetric shrinkage achieved was 11.68% whereas when using conformal cooling system, the volumetric shrinkage achieved was 11.59%.This is because the conventional cooling system covers exterior part of the product only whereas the conformal cooling channel covers exterior and interior of the product.

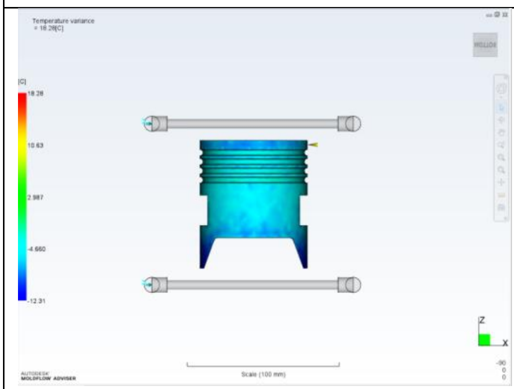
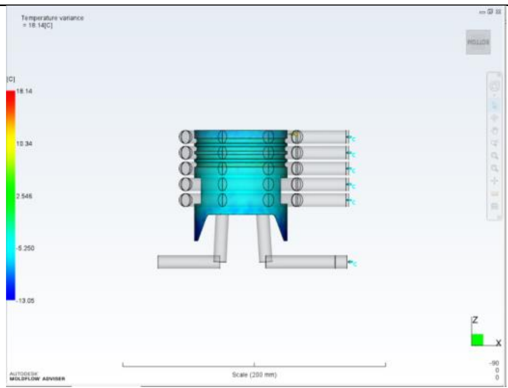
Conventional cooling system	*Conformal cooling system*
	

Figure 7: Comparison of cooling system

When using the conventional cooling system, the temperature variance achieved was 18.28°C whereas when using conformal cooling system, the temperature variance achieved was 18.14°C. The minimum temperature variance results in minimum warpage.

Conventional cooling system	*Conformal cooling system*

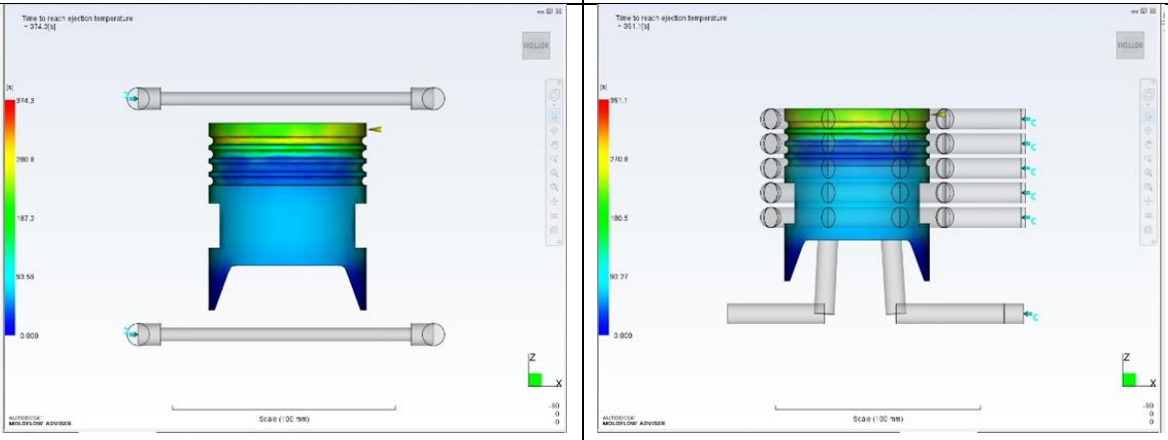


Figure 8: Comparison of cooling system

When using the conformal cooling system, the time taken to reach ejection temperature is 361.1s. While, using the conventional cooling system, the time taken to reach ejection temperature is 374.3s. Thus, as using conformal cooling system, the cycle time of a part can be reduced with minimum warpage.

3.3 Discussions

The result from each analysis was compared to choose the best design where the process parameters are contributing to high quality of mould insert and less causes less defects. Then this design has been created in mould insert.

Table 1 – Result of gate location analysis

Parameter	Gate location A	Gate location B
Fill time	4.06s	4.56s
Quality predication	High, medium	High, medium
Time taken to reach ejection temperature	373.1s	429.2s
Air traps	Less compare to B	More compare to A
Weld lines	Less	More
Confidence to fill	High	High
Injection Pressure	4.83	4.21

Table 2 – Result of cooling channel analysis

Parameter	Conventional Cooling	Conformal Cooling
Temperature Variance	18.28ºC	18.14ºC
Volumetric Shrinkage at ejection	11.68%	11.59%
Time taken to reach ejection temperature	374.3s	361.1s

The table shows the yellow colour that represent the good condition of the design. Gate location A have a lot of positive data compared to gate location B. The cooling channel effects the part quality to better than without cooling channel. Even though the quality prediction for gate location B is better than gate location A, other parameters such as fill time, weld line, air traps and ejection temperature duration is better for gate location A. Thus, gate location A is chosen over gate location B.

4. Conclusion

In this paper, an optimum gate location is determined. There are two different trials done on type of gate to be used for this product. Out of these two trials, it is observed from Figure 4 that location gate is optimum gate for this product. With use of location gate, A, the fill time is 4.06 s and it has minimum weld line and air traps, also there are no issues of quality and de-gating.

There are two different types of cooling systems being analysed. It is observed from Table 2 that Conformal Cooling System is the ideal cooling system for this product. It leads to better cooling properties due to exhibiting the lower volumetric shrinkage. It also provides the lower time to reach the ejection temperature, which translates to lower cooling time and reduced overall cycle time. The conformal cooling channel shows near uniform cooling that makes it most favourable cooling system.

So, use of an injection moulding analysis software provides valuable information for plastic product and mould design in reducing time and cost of production especially for complex parts.

Acknowledgement

This research was made possible by support from the Ministry of Higher Education, Malaysia. The authors would also like to thank the Faculty of Mechanical Engineering and Manufacturing, Universiti Tun Hussein Onn Malaysia for their support.

References

[1] M. Scholar and M. Koli, "Cycle Time Optimization in Injection Moulding." 2019.

[2] H. Hassan, N. Regnier, C. Lebot, C. Pujos, and G. Defaye, "Effect of cooling system on the polymer temperature and solidification during injection molding," *Appl. Therm. Eng.*, vol. 29, no. 8–9, pp. 1786–1791, 2009, doi: 10.1016/j.applthermaleng.2008.08.011.

[3] H.-S. Park and X.-P. Dang, "Development of a Smart Plastic Injection Mold with Conformal Cooling Channels," *Procedia Manuf.*, vol. 10, pp. 48–59, 2017, doi: https://doi.org/10.1016/j.promfg.2017.07.020.

[4] O. Mohamed, "A Simulation Study of Conformal Cooling Channels in Plastic Injection Molding," *Int. J. Eng. Res.*, vol. 2, pp. 344–348, 2013.

[5] M. Khan, S. K. Afaq, N. U. Khan, and S. Ahmad, "Cycle Time Reduction in Injection Molding Process by Selection of Robust Cooling Channel Design," *ISRN Mech. Eng.*, vol. 2014, p. 968484, 2014, doi: 10.1155/2014/968484.