Analysis of coolant temperature and injection time effect on the product quality in injection moulding using the Finite Element Method (FEM)

Revangga Anjar Achmaed Pratama¹, Aminnudin*¹, Yanuar Rohmat Aji Pradana¹, Nurul Afifah¹
¹Mechanical Engineering Department, State University of Malang
Jl Semarang no 5, Malang, Indonesia
Corresponding Autor : Aminnudin
Email: aminnudin.ft@um.ac.id

Abstract. The purpose of this research is to analyze the effect of cooling temperature and injection time on the injection molding machine to product quality using the finite element method. The first step in this research a designing cavity molding and plastic product with AUTODESK INVENTOR 2019 PROFESSIONAL software. The next step is found the average temperature in molding use the ANSYS software. The results are then used for the process of analyzing the quality of plastic products. The next stage is the process of analyzing the quality of plastic products using AUTODESK MOLDFLOWADVISER R2 2017. The plastic product analyzed is a product called plug-ib, the product used in eye drops that are obtained from plastic manufacturing companies, namely PT. Berlina Tbk. The results obtained by a cooling temperature of 24 °C with an injection time of 1.4 seconds obtained the quality of the best product with a percentage of product quality of 79% and a product weight of 0.415 grams.

Keywords: injection molding, finite element method, molding

1. Introduction
Plastics are polymers that have unique and extraordinary properties. The polymers are a material consisting of molecular units called monomers [1][2][3]. In the Republic of Indonesia, Indonesia provides 925 manufacturing industries which produce 4.6 million tons per year to meet the demand for national plastic products [2]. One of the plastic manufacturing processes uses an injection molding machine.

Injection molding is a plastic production processed by injection molding machine [4] [5]. The most influence to the quality of the product on the injection molding machine is the use of coolant temperatures and injection time setting[6] [7]. Coolant temperature will affect the product shrinkage. To get the minimum shrink mark, we must use a low cooling temperature [8] [9]. Injection time can affect the weight of the product, so the percentage of product weight difference that occurs is a minimum of 0.35% and a maximum of 1.43% [6]. In the Republic of Indonesia, to get good product quality still using trial and error product processes which can lead to wasteful use of plastic materials [10]. This study uses FEM to look for product quality in injection molding machines with variations in coolant temperature and injection time to get the optimal value as an injection molding machine set.
2. Material and Methods

Material used in this research is LDPE (low density Polyethylene). LDPE has melting point 110 °C, specific density 0.925 g/cm³, and thermal conductivity 0.4 [11] [12][13]. This research begins with the molding part design process, namely cavity molding and plastic parts or plastic products called Plug-Ib, this product is a plastic product for eye drops produced by PT. Berlina Tbk as shown in the Figure 1. For the design process using the 2019 AUTODESK INVENTOR PROFESSIONAL software.

![Figure 1. a) Cavity Molding b) Product](image)

The variation of coolant temperatures used for the research process was 18, 22, and 24 °C as well as the variation in the injection time used for the study of 1.4; 1.6; and 1.8. The next step is knowing the average temperature of cavity molding using ANSYS software. To determine the average temperature used Thermal Transient on ANSYS toolbox.

The first step is to enter the cavity molding geometry model, then the meshing process is carried out in this step also set boundary conditions such as steps, initial temperature, analysis settings, and solutions. The next step is the running process to get results. This research process focused on the product cavity in cavity molding as shown in Figure 2. This is done because the process of product formation is in the product cavity in cavity molding.

![Figure 2. The Analysis Locate Focus in the Cavity Molding](image)

The next step is to analyze the quality product using the AUTODESK MOLDFLOW ADVISER R2 2017 software. The step for the analysis process is the results of the average temperature from ANSYS included in the wizard setting in the form of temperature molding. The setting process that
is carried out before the analysis is temperature molding, injection time, injection pressure, temperature melt, material, injection location, and accuracy. Results from the MOLDFLOW is filling time, gate pressure, quality prediction, weld lines, cooling products, defects, and shrink marks.

3. Results and Discussion

Colour plot thermal flow of melt in the cavity molding can be seen in Figure 3. Molding with a cooling temperature of 18 °C produces an average temperature in the product cavity of 43.6 °C. When the cooling temperature is increased to 22 °C, the average temperature is 47.1 °C. When the molding cooling temperature is increased to 24 °C the average temperature is 48.9 °C.

![Figure 3. Colour Plot Thermal flow of Melt in the Cavity Molding](image)

3.1 Fill Time

The product fills time or melts time into the product cavity as shown in the Figure 4. Fill time or the time the melt enters the product cavity is different from the injection time. This fill time is affected by the gate hole size in the cavity, injection time, cooling process, and viscosity.

![Figure 4. Colour Plot of Fill Time in the Product](image)
From the results of the analysis process, it is obtained that the long fill time product is 1.95 seconds with a coolant temperature variation of 18 °C and injection time for 1.8 seconds, while the fastest is 1.5 seconds occurs at a cool temperature of 18 °C and time injection for 1.4 seconds.

3.2 Gate Pressure
Gate pressure is the pressure that occurs at the gate when the melt enters the cavity (Figure 5). Gate pressure is influenced by injection time, if the injection time is getting faster than the gate pressure drops quickly this is because the melt that is injected experiences a rapid pressure drops because the time to push the plastic is short [7].

From the analysis process, it is found that the gate pressure which decreases rapidly occurs at a coolant temperature of 18 °C and the injection time for 1.4 seconds get the pressure of 2.681 MPa, it is caused by the temperature at a low molding, while the largest is 3,175 MPa which occurs at a coolant temperature of 24 °C and injection time for 1.4 seconds caused by a short injection time the melt that enters first will experience an early solidification so that the gate hole has a slight blockage.

3.3 Quality Prediction
Quality prediction is the result of predicting the quality of the product as shown in the Figure 6. The quality prediction parameters are filled time, temperature, and ejection time [8]. The product analysis process that has been carried out gets the highest product quality prediction percentage that is 79.5% and the smallest is 59.3%. The highest percentage is at a molding temperature of 24 °C with an injection time of 1.4 seconds producing a prediction percentage of product quality of 79.5%, while the smallest percentage at a molding temperature of 18 °C with an injection time of 1.4 seconds is equal to 59.3%.

![Figure 5. Colour Plot of Gate Pressure in the Product](image)

![Figure 6. Colour Plot of Quality Prediction in the Product](image)
This study found no low-quality levels but found a medium-quality level. Medium-quality is still a tolerance if the product is produced. The causes of medium level quality products are lack of injection pressure and cooling that is too fast or slow, resulting in product defects. The product analysis data does not present medium-level quality prediction percentage data because this research is looking for good quality seen from the high-quality level of the product.

The analysed product has a weld line angle of 0.2443° as shown in the Figure 7. The biggest weld line angle is 135°, so it is predicted to experience product defects in the form of short shots. The cause of this weld line is because the Plug-Ib product has 2 points injection or multi-point gate injection.

![Figure 7. Weld Line in the Product](image)

### 3.4 Cooling Time Product

The simulation results obtained the fastest cooling time of 1.4 seconds with a molding temperature of 43.6 °C caused by a low mold cooling temperature and fast injection time. The results of the analysis with an injection time of 1.8 seconds with a molding temperature of 43.6; 47.1; and 48.9 °C obtained the longest product cooling time data is 1.475 seconds (Figure 8). The long product cooling time is due to the long injection time, so the product forming or solidification process requires a long cooling time.

![Figure 8. Colour Plot of Cooling Time in the Product](image)

### 3.5 Product Defect

The types of product defects that occur are a bubble and short shot as shown in the Figure 9. Short shots that occur are not large or affect the dimensions of the product. The location of the product defect in the form of a short shot is found on the product press. Short shot occurs because the product fill time is too low, so there are parts that are short or do not contain.
Bubble or air trapped in the product is caused when the product is cooling too fast as shown in the Figure 10. The molding coolant temperature is 18 °C with an injection time of 1.4 seconds, there is a high point of bubble formation. The cause of the differences in the many points of the bubble that is, the first analysis data found that the cooling time is too fast so that many bubble points occur.

3.6 Defect Volume
From the simulation results using the AUTODESK MOLDFLOW software, it is found that the volume of disability or there is still parts that have not been filled by plastic material during the injection process with the largest defect volume is 0.00116 cm³ and the lowest is 0.0006 cm³. The simulation results show that the injection time setup for 1.4 seconds with molding templates of 43.6 °C has the greatest volume of disability because of the presence of numerous defect points in the form of bubbles. Whereas at a molding temperature of 47.1 °C with an injection time of 1.8 seconds it has a disability volume that is slightly compared to other setups.

3.7 Product Weight
Before the analysis process is carried out, it is known that the weight of the product in the field is 0.43 gram, after the analysis process, the weight of the product is decreased by 0.389 to 0.415 gram
because there is a product defect, so the weight of the product is different in PT. Berlina Tbk. The difference in weight of plug-Ib products is because in the field Plug-Ib products use pure LDPE material mixed with aflag or recycled material.

3.8 Product Shrinkage or Shrink Mark

Figure 11 shows colour plot of shrink mark in the tube product. Simulation results show the percentage of depreciation of 0.08% or not exceeding 1%. But the value of the depth of shrinkage in different products, the greatest depth is 0.0079 mm and the lowest is 0.006 mm. The cause of product shrinkage of 0.0079 mm is the coolant temperature in molding that is too cold that is equal to 18 °C with an injection time of 1.4 seconds so that when the melting process enters the mold and cooling is too fast. The lowest product shrinkage depth of 0.006 mm is found in the molding cooling temperature of 24 °C with an injection time of 1.4; 1.6; and 1.8 seconds.

4. Conclusion

The results of this study are with a coolant temperature of 24 °C with an injection time of 1.4 seconds, we get the best quality of the products from all simulation results to find product quality with different variations in coolant temperature and injection time. In this research, the temperature of the refrigerant is very influential on the quality of the product so that it can result in product defects in the form of shrink marks and bubbles, whereas if it is wrong in the injection time setting, the product will experience defects in the form of short shots.

5. Reference

[1] Fatima M, Abderazzak O, Mohamed B and Mohamed E, 2017 Mechanical behavior prediction of PPR and HDPE polymers through newly developed nonlinear damage-reliability models Procedia Struct. Integr. 3 387–94
[2] Yusup E M, Mahzan S and Kamaruddin M A H, 2019 Natural Fiber Reinforced Polymer for the Application of Sports Equipment using Mold Casting Method IOP Conf. Ser. Mater. Sci. Eng. 494
[3] Solechan, Rubijanto J P, Triyono J and Pujiyanto E, 2019 Study of Making Implant Plate and Screen of Femur Bone Internal Fictation from Hydroxyapatit Bovine and Polymer Biodegradation Material Using 3D Printers on Mechanical Strength IOP Conf. Ser. Mater. Sci. Eng. 494
[4] Tran N T and Gehde M, 2019 Creating material data for thermoset injection molding simulation process Polym. Test. 73 284–92
[5] Lay M, Thajudin N L N, Hamid Z A A, Rusli A, Abdullah M K and Shuib R K, 2019 Comparison of physical and mechanical properties of PLA, ABS and nylon 6 fabricated using fused deposition modeling and injection molding Compos. Part B Eng. 176 107341
[6] Cao W, Shen Y, Wang P, Yang H, Zhao S and Shen C, 2019 Viscoelastic modeling and simulation for polymer melt flow in injection/compression molding J. Nonnewton. Fluid Mech. 274 104186
[7] Kumar B P, Venkataramaiah P and Ganesh J S, 2019 Optimization of process parameters in injection moulding of a polymer composite product by using gra Mater. Today Proc. 18 4637–47
[8] Stryczek J, Banaś M, Krawczyk J, Marciniak L and Stryczek P, 2017 The Fluid Power Elements and Systems Made of Plastics Procedia Eng. 176 600–9
[9] Li L, Sun L, Dai Z, Xiong Z, Huang B and Zhang Y, 2019 Experimental investigation on mechanical properties and failure mechanisms of polymer composite-metal hybrid materials processed by direct injection-molding adhesion method J. Mater. Process. Technol. 263 385–95
[10] Ni’am L, Budiwantoro C and Rahman M, 2017 Desain dan Optimasi Injection Mold Sistem Slider pada Produk Stick T15 J. Rekayasa Mesin 8 155–65
[11] Chaoanchaikul K, Kositchaiyong A and Sombatsompop N 2009 Blending Techniques Affecting Mechanical and Morphological Properties of Fly Ash / LDPE and CaCO 3 / LDPE Composites 17 281–91
[12] Chen Y, Wan C, Liu T, Chen Z, Qiao Y, Lu J, Yan J, Zhao L and Esseghir M, 2018 Evaluation of LLDPE/LDPE blend foamability by in situ rheological measurements and bubble growth simulations Chem. Eng. Sci. 192 488–98
[13] Callister W W D J, Rethwisch D G D, Callister. William D, Callister W W D J and Rethwisch D G D, 2007 Materials science and engineering: an introduction (Danver: John Willey and Sons)