Simulation of melt temperature on quality of Pepsodent toothbrush products in injection molding process using Ansys software

Arief Susetyo, Andoko*
Mechanical Engineering Department, State University of Malang
Semarang 5, Lowokwaru, Malang, Indonesia
*Email: andoko.ft@um.ac.id

Abstract. Shrinkage is a defect in the form of a change in the dimensions of the product as a result of the injection molding process. Cooling mold is one of the factors that influence shrinkage product defects. Cooling the mold is done using a fluid medium (water cooling). The test was carried out using polypropylene material with constant parameter set-up such as injection time of 2 seconds, injection pressure of 10 MPa, coolant temperature of 22, 24, and 26 °C and melt temperatures of 190, 200 and 210 °C. Simulation uses Ansys transient software using the finite element method to analyze the average temperature in the product cavity. Simulation was using Pepsodent toothbrush products. Simulation result shows the best product quality with a product quality value produces a fill time value of 3.19 seconds with a predictive quality of 68.5% and has a weld line of 0.0115 °, a cooling time of 13.8 seconds and a shrinkage value of 0.008 mm. Simulation results show that the optimal product quality values are found in the set-up parameters of 24 °C coolant temperature, 200 °C melt temperature and 41.3 °C mold temperature. Product quality safety factor indicates that the product is in a good category. This means that the value of product quality is influenced by the mold cooler temperature and along with the increase in the mold temperature, which causes the mold cooling temperature is not constant.

Keywords: shrinkage, injection molding, finite element method

1. Introduction
Plastic is a synthetic material that can be deformed and can be maintained and hardened by adding other materials in a composite to it. When exposed to heat and pressure, the material formed from this polymer material can be formed into any desired shape using the injection molding process [1]. Plastic materials used in the manufacture of plastic products include polypropylene, polyethylene, polystyrene, and others. Polypropylene is used in a variety of applications, such as automotive components, laboratory equipment, food or beverage containers [2].

Injection molding process is a method used to produce complex geometrically shaped products with high productivity and accuracy and at low cost [3]. Plastic production process using an injection molding machine or using a blow molding machine, there are 4 main factors that can affect product quality, namely plastic material selection, machine setting process, product design
and molding design [4]. Cooling system in injection molding process affects the quality of the plastic product, if mold is too hot then melt or liquid plastic will experience a slowdown in freezing process resulting in defects in product, whereas if it is too cold product will freeze too quickly and shape of product is imperfect. Designer must design a cooling system that can cool in between 70% - 80% of one injection system cycle and temperature difference must not exceed 25% in order to get good cooling results in mold [5].

Molding cooling system has several types of cooling flow, a good cooling flow type, namely turbulent flow type because this flow can dissipate heat on the molding wall better than laminar flow [6]. Molding design process that must be considered is use of type cooling circuit, thickness of cooling walls, distance of cooling holes and diameter of the cooling holes [7]. Cooling temperature in the molding will affect shrinkage of the product or shrinkage to get the minimum shrinkage, so the optimal cooling temperature must be used [8]. Melt temperature or liquid plastic is temperature at which material begins to change from solid to melt. Basically, higher temperature of liquid plastic higher the process temperature. In plastics industrial applications, melting temperature is used to identify plastic material [3]. Plastic production process to get right injection machine parameters is carried out trial and error, because in injection molding there is no standardized trial and error product data [9].

Research is a development of previous research, previous researchers examined shrinkage defects in polypropylene products using cooling tower media and air using experimental methods. The purpose of study is to examine product quality and product shrinkage defects with variations in cooling temperature and melt temperature in injection molding process using software based on finite element method.

2. Method

Simulation analyzes the effect of coolant temperature and melt temperature using Ansys Workbench 18.1 software. Cooling temperature in molding will affect the shrinkage of product or shrinkage to get the minimum shrinkage, so you must use the optimal cooling temperature [8]. Temperature of melt or liquid plastic is the temperature at which the material begins to change from solid to melt. Higher the temperature of the liquid plastic, the higher the process temperature [3]. High temperatures are used to accelerate filling of material into mold cavity because the higher temperature the lower viscosity, but this has an impact on greater product shrinkage [10].

2.1 Design

Pepsodent toothbrush products in this article were designed using Autodesk Inventor Professional 2018 software with following dimensions as a mold. Pepsodent toothbrush uses metal steel material. Material properties consist of density = 7850 kg m\(^{-3}\), modulus of elasticity = 200 GPa, thermal conductivity = 60.5 W m\(^{-1}\) K\(^{-1}\), specification of heat = 434 J kg\(^{-1}\) K\(^{-1}\) and coefficient of thermal expansion = 1.2e \(x\) 005.

2.2 Simulation Ansys

Results of pepsodent toothbrush product design were simulated with Ansys workbench 18.1 software using Finite Element Method. Finite Element Method is used to analyze average temperature in cavity. Finite element method analysis provides a way to carry out easy and efficient research on a wide range of used parameters with easily evaluated design and manufacturing conditions [11-14].

Simulation on cavity part is carried out by the analysis process to determine the average temperature at the time the molten plastic is injected into the cavity. Results of analysis carried
out obtained mold temperature as basis for analyzing quality of pepsodent toothbrush product. Boundary conditions applied in this simulation are geometric boundary conditions, material properties and loads.

2.3 Simulation Autodesk Moldflow Adviser
Simulations were carried out on pepsodent toothbrush products to determine product quality using Autodesk Moldflow Adviser 2017. Pepsodent toothbrush simulation process is to provide various maximum conditions, including providing variations in mold temperature, melt or liquid plastic temperature, injection pressure and injection time.

3. Results and discussion
Results of analysis of plastic products or parts on variations in coolant temperature and melt temperature each have 9 different data. After analysis process is carried out using Ansys software, following data are obtained.

| No | Cooling Temperature (°C) | Melt Temperature (°C) | Mold Temperature (°C) |
|----|------------------------|----------------------|----------------------|
| 1  | 22                     | 190                  | 37.8                 |
| 2  | 22                     | 200                  | 38.5                 |
| 3  | 22                     | 210                  | 39.8                 |
| 4  | 24                     | 190                  | 40.9                 |
| 5  | 24                     | 200                  | 41.3                 |
| 6  | 24                     | 210                  | 42.5                 |
| 7  | 26                     | 190                  | 43.6                 |
| 8  | 26                     | 200                  | 45.9                 |
| 9  | 26                     | 210                  | 47.3                 |

Figure 1. Meshing design geometry
Figure 1 shows meshing of product design geometry. Meshing is performed to determine value at each product geometry point. This process produces a mesh with 75927 nodes and 49740 elements. Size of mesh contained in an object will affect accuracy of the FEM analysis to be carried out. Smaller mesh size on an object, more accurate results will be. Figure 2 shows thermal flow in cavity molding section. Part of cavity that has the highest temperature is at gate location cavity or the entry hole for liquid plastic into product cavity in cavity. Molding temperature analysis is focused on product cavity. Product cavity is main part of forming a plastic product or part. Presentation of results of mold temperature analysis shows an increase in temperature because cooler temperature used is getting bigger. Mold temperature required to print a PP (Polypropylene) thermoplastic is a minimum of 20 °C and a maximum of 80 °C [15]. Temperature probe is used during analysis process at Ansys to find out the mold temperature in product cavity. Analysis is focused on product cavity, because product cavity is main place for product forming process and place for heat transfer from molten plastic to cavity molding part.

Table 2. Data Fill Time, Quality Prediction, Weld Line, and Cooling Time

| No | Temperature Cooling (°C) | Temperature Melt (°C) | Fill Time (S) | Quality Prediction (%) | Weld Line (°) | Cooling Time (S) |
|----|--------------------------|-----------------------|---------------|------------------------|--------------|-----------------|
| 1  | 22                       | 190                   | 3.165         | 64.2 H                 | 0.7139°      | 13.5            |
| 2  | 200                      | 3.174                 | 64.3 H        | 0.4918°                | 13.55        |
| 3  | 210                      | 3.184                 | 64.4 H        | 0.3826°                | 13.55        |
| 4  | 190                      | 3.187                 | 64.5 H        | 0.0143°                | 13.58        |
| 5  | 200                      | 3.192                 | 68.5 H        | 0.0115°                | 13.58        |
| 6  | 210                      | 3.195                 | 64.5 H        | 0.0143°                | 13.59        |
| 7  | 190                      | 3.195                 | 64.3 H        | 0.2543°                | 13.59        |
| 8  | 200                      | 3.212                 | 64.3 H        | 0.9948°                | 13.62        |
| 9  | 210                      | 3.212                 | 64.5 H        | 1.165°                 | 13.64        |
Table 3 shows product analysis result data obtained from simulation with Autodesk Moldflow Adviser software. Figure 3 shows simulation data results of pepsodent toothbrush handle products. In the table it can be seen that the value of the fill time or time of liquid plastic has increased at 24 and 26 °C cooling temperatures. The increase in time is caused by cooling molding process and increasing temperature of melt or liquid plastic. Fill time also affects quality of prediction product. Parameters that affect quality prediction results are fill time, temperature and ejection time [5]. Simulation results obtained that highest quality prediction is 68.5% and lowest is 64.0%. Percentage of imperfect product quality predictions is due to the fact that there are still spots of bubble or air voids in product. Weld line that was produced during injection process was not able to close completely. Product cooling time is length of time used to cool product after cavity is completely filled with liquid plastic. Product cooling times that are too fast are likely to experience shrinkage defects and bubbles [9].

3.1 Shrinkage product
Shrinkage is a defect in form of a change in dimensions of product resulting from injection molding process. Product Shrinkage or Shrinkage is due to differences in thickness so that during solidification plastic material will shrink [16]. Results of the analysis process carried out by the author are very small product shrinkages (Figure 4).

Figure 3. Simulation data of toothbrush handle

Figure 4. Shrinkage in toothbrush handle
Table 3. Data of product shrinkage

| No | Cooling Temperature (°C) | Melt Temperature (°C) | Percentage of Shrinkage (%) | Depth of shrink (mm) |
|----|--------------------------|-----------------------|------------------------------|---------------------|
| 1  | 22                       | 190                   | 1%                           | 0.041               |
| 2  | 200                      | 200                   | 1%                           | 0.017               |
| 3  | 210                      | 210                   | 1%                           | 0.016               |
| 4  | 24                       | 190                   | 1%                           | 0.011               |
| 5  | 200                      | 200                   | 1%                           | 0.008               |
| 6  | 210                      | 210                   | 1%                           | 0.011               |
| 7  | 26                       | 190                   | 1%                           | 0.012               |
| 8  | 200                      | 200                   | 1%                           | 0.081               |
| 9  | 210                      | 210                   | 1%                           | 0.092               |

Table 3 shows that the percentage of shrinkage is 1%. However, depth of shrinkage of product is different on average, with largest depth being 0.092 mm and smallest being 0.008 mm. What causes product shrinkage of 0.092 mm is cooling temperature in molding which is 26 °C with a liquid plastic temperature of 210 °C and a mold temperature of 47.3 °C, this is due to inadequate cooling and temperature of liquid plastic which is too hot so that when liquid plastic process fills the mold, product has a delay in product formation [3]. While the smallest shrinkage depth of 0.008 mm is found at a cooling temperature of 24 °C with a liquid plastic temperature of 200 °C and a mold temperature of 41.3 °C. With optimal cooling temperature and temperature of liquid plastic that is not too hot so that solidification process or product formation is not too fast so that parts of product with a small volume of plastic can still be filled. Figure 5 shows location of shrinkage on product. Simulation results of Autodesk Moldflow Adviser software provide information from all product simulation experiments in good condition because from simulation data product has a shrinkage percentage of plastic material below 1% so that if the product is produced the shrinkage that will occur will not damage dimensions of product itself.

4. Conclusions
Analysis of pepsodent toothbrush product simulations was carried out based on the loading determined in each pepsodent toothbrush product simulation. Results of simulation analysis on pepsodent toothbrush products obtained optimal set-up parameters to produce good product quality, namely the set-up parameters of 24 °C coolant temperature, 200 °C melt temperature and 41.3 °C mold temperature, this means that the product quality is not good. Occurs because the increase in mold temperature causes the cooling temperature that occurs in the mold to have a temperature that is not constant and will have an impact on the amount of shrinkage value that is getting deeper.

5. References
[1] R. A. Malloy, Plastic Part Design for Injection Molding. 2010.
[2] S. Ersoy, “Plastic Injection Machine,” 2014.
[3] H. Hassan, N. Regnier, C. Lebot, “Effect of cooling system on the polymer temperature and solidification during injection molding,” Appl. Therm. Eng., vol. 29, no. 8–9, 2009.
[4] J. G. Kovács and B. Sikló, “Investigation of cooling effect at corners in injection molding,” Int.
Commun. Heat Mass Transf., vol. 38, no. 10, pp. 1330–1334, 2011.

[5] M. Jamsheed and Rahman, “Design and Analysis of Plastic Injection Mould for CAM BUSH with Submarine Gate,” Mater. Today Proc., vol. 2, no. 4–5, pp. 2083–2093, 2015.

[6] D. O. Kazmer and D. O. Kazmer, Injection Mold Design Engineering. 2016.

[7] S. J. Park and T. H. Kwon, “Optimal cooling system design for the injection molding process,” Polym. Eng. Sci., vol. 38, no. 9, pp. 1450–1462, 1998.

[8] J. Sreedharan and A. K. Jeevanantham, “Analysis of Shrinkages in ABS Injection Molding Parts for Automobile Applications,” Mater. Today Proc., vol. 5, no. 5, pp. 12744–12749, 2018.

[9] C. L. Li, C. G. Li, and A. C. K. Mok, “Automatic layout design of plastic injection mould cooling system,” CAD Comput. Aided Des., vol. 37, no. 7, pp. 645–662, 2005.

[10] Y. Liu and M. Gehde, “Evaluation of heat transfer coefficient between polymer and cavity wall for improving cooling and crystallinity results in injection molding simulation,” Appl. Therm. Eng., vol. 80, pp. 238–246, 2015.

[11] A. Andoko, P. Paryono, R. Prasetya, R. P. Jeadi, P. Kurniawan, and D. R. Pradica, “Simulation of the effect of energy absorption on crushbox with full crush initiator and without crush initiator,” in AIP Conference Proceedings, 2020, vol. 2262.

[12] Andoko and P. Puspitasari, “Characteristics of leaf spring strength of material 65Si7 and material C17000 using finite element method,” AIP Conf. Proc., vol. 1778, no. October, 2016.

[13] A. Andoko et al., “The influence of crush initiator placement towards the application of energy crash box due to impact load,” in AIP Conference Proceedings, 2020, p. 040016.

[14] A. Andoko et al., “Simulation on CNC 5 axis milling spindle bolt using finite element method,” in AIP Conference Proceedings, 2020, vol. 2262.

[15] T. Kanai and G. A. Campbell, Polymer Processing Society Progress in Polymer Processing Series. 2014.

[16] M. A. M. Amran, N. Idayu, K. M. Faizal, M. Sanusi, R. Izamshah, and M. Shahir, “Part weight verification between simulation and experiment of plastic part in injection moulding process,” IOP Conf. Ser. Mater. Sci. Eng., vol. 160, no. 1, 2016