JOURNAL OF APPLIED MECHANICAL ENGINEERING INNOVATION

Journal homepage: https://journal.uny.ac.id/publications/jamei

The Effect of Temperature and Pressure on Filling Time in the Injection Process of Shop Hanger Products

Thedy Kristanto [a,*], Paryanto [b], Heri Wibowo [a]

[a]*Department of Mechanical and Automotive Engineering, Universitas Negeri Yogyakarta, Indonesia*
[b]*Department of Mechanical Engineering Education, Universitas Negeri Yogyakarta, Indonesia*

Corresponding author: TheydKristanto2@gmail.com

ARTICLE INFO

Article history:

Received 30.08.2025
Revised 15.09.2025
Accepted 07.10.2025

Keywords:

filling time, injection molding, pressure, temperature

ABSTRACT

This study investigated the effect of melt temperature and injection pressure on filling time in the injection molding process for shop hanger products. The research used an experimental simulation method with Autodesk Moldflow Adviser 2023. The melt temperature was varied at 180, 190, 200, 210, and 220 °C, while injection pressure was varied at 20, 30, 40, and 50 MPa. Runner diameters of 2, 4, and 6 mm were also evaluated to determine the most suitable flow channel for the mold design. The results show that a 4 mm runner produced the shortest filling time among the runner variations. Increasing temperature and pressure tended to reduce filling time within the tested parameter range. The shortest filling time was obtained at a melt temperature of 220 °C and an injection pressure of 40 MPa with a 4 mm runner. Several defects, including short shot, warpage, weld lines, sink marks, and air traps, were detected during the simulation, but the defects were not significant under the optimal parameters. The mold was designed with a 330 × 350 mm mold base and a required clamping force of 470 kN. The selected injection machine available at CV. Mulus Teknik, a 90-ton UN90SKII machine with a clamping force of 900 kN, was suitable for production.

1 Introduction

Technology is an essential aspect of current development. Around us, technology continues to advance from Industry 1.0 to Industry 4.0. The purpose of these developments is to meet human needs and improve labor efficiency; therefore, the development of current technology is particularly important in the manufacturing industry.

Manufacturing companies play an important role in the Indonesian economy because their main processes convert raw materials, components, and other materials into finished products that meet specified standards. In production, most manufacturing companies already use machines and advanced equipment to accelerate production processes.

According to the Ministry of Industry of the Republic of Indonesia, in 2022 the manufacturing sector consistently remained above the expansion level. In December 2022, Indonesia's Manufacturing PMI reached 50.9, higher than the previous month at 50.3. Based on the S&P Global survey, Indonesia's Manufacturing PMI had remained in an expansionary phase for approximately sixteen consecutive months since September 2021. This positive performance indicates that national manufacturing continued to recover after the Covid-19 pandemic despite uncertain global economic conditions and recession risks.

Plastic-based products are common in daily life and create opportunities for businesses to manufacture goods used every day. Injection molding is frequently used to produce plastic products because it can create complex geometries and production characteristics that are difficult to obtain using other techniques [1], [2].

In plastic product molding, process parameters such as temperature, pressure, injection time, and cooling time must be considered because they strongly influence melt flow, product quality, shrinkage, and defect formation [3], [4]. If these parameters are not suitable, plastic injection results may be poor, including incomplete shapes, product shrinkage, dimensional deviation, and cracking. These defects increase material waste and production cost. Defects in plastic products often occur because of unsuitable parameters, particularly injection temperature and pressure. Temperature is required to melt the plastic so that it can fill the die cavity, while pressure is the force required to inject the molten plastic into the mold [5], [6].

This research focuses on Indonesian manufacturing industries, particularly small and medium enterprises and technical workshops. Along with technological development, many SMEs in Indonesia still face difficulties in adopting technologies used by large industries. Large industries have commonly implemented pre-production simulation to minimize excessive costs and reduce the risk of mold-design errors before production [7], [8], [9], [10].

This study was conducted at CV. Mulus Teknik in Bandung during the production of a molding die for shop hanger products. The main problem was determining appropriate temperature and injection pressure parameters during trial production. The final objective was to evaluate the effect of temperature and pressure variations on filling time for polypropylene material and to determine suitable mold and machine requirements for shop hanger production.

2 Method

This research used an experimental simulation method. Experimental research is a method used to determine the effect of a particular treatment on another variable under controlled conditions [10]. The study analyzed the filling time of a new shop hanger product design by varying temperature and pressure to obtain suitable parameters for mold production. A literature review and a three-dimensional product model were used as the basis for geometry preparation and analysis. Temperature and pressure simulations were carried out using Autodesk Moldflow Adviser 2023 to determine their effect on filling time, possible defects, and the mold design for the shop hanger product.

2.1 Research Procedure

The research procedure began with a literature study related to mold production units and production limitations as a reference for designing the shop hanger mold. In the mold design process, several aspects were considered, including material selection, product complexity, product dimensions, and the determination of the core and cavity according to CV. Mulus Teknik standards. The next stage was product design, which included concept preparation, development, and refinement to solve the production problem. The shop hanger product was designed using Autodesk Inventor 2022 based on customer requirements at CV. Mulus Teknik.

After the product design was completed, runner size selection was carried out by testing runner diameters of 2, 4, and 6 mm using Autodesk Moldflow Adviser 2023 to determine the optimal runner size. The effects of melt temperature and injection pressure on filling time were then analyzed using the same software. This simulation aimed to determine the ability of the material to fill the product cavity and to identify possible defects during the injection molding process. Furthermore, the mold construction design was developed based on a two-plate mold base in Autodesk Inventor 2022, with the mold base size adjusted to the availability at CV. Mulus Teknik. The final stage of the research was drawing conclusions from all simulation and design results to determine suitable injection temperature and pressure parameters, as well as the machine requirements for shop hanger production.

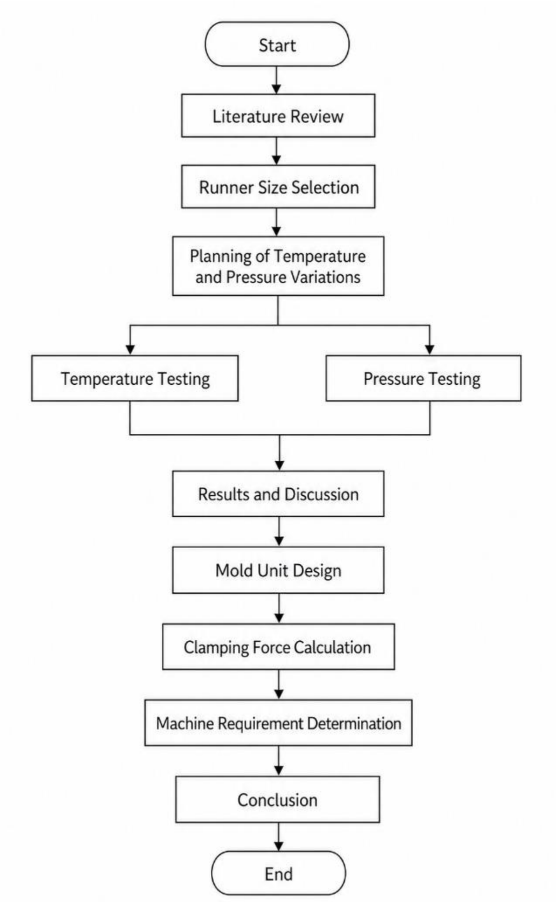

Figure 1. Research flowchart.

2.2 Tools, Materials, and Data Collection

The product and mold construction were designed using an Asus VivoBook laptop and Autodesk software. Product planning used Autodesk Inventor 2022, while the simulation used Autodesk Moldflow Adviser 2023.

Table 1. Computer specifications

Specification	Description
Operating system	Windows 11
Processor	Intel
CPU	Core i5-8265U
Memory	4096 MB
VGA	Intel® UHD Graphics 620

Data were collected through simulation tests by varying melt temperature and injection pressure. The temperature variations were 180, 190, 200, 210, and 220 °C. The injection pressure variations were 20, 30, 40, and 50 MPa. The data analyzed were filling time and defects detected in the simulated products. The simulation results were used as a reference for determining the mold design and machine requirements.

3 Result and discussion

3.1 Product Identification

Mold design requirements were identified from product information, mold requirements, and polypropylene material properties. The product specifications are presented in Table 2, mold requirements in Table 3, and polypropylene properties in Table 4.

Table 2. Product specifications

Item	Specification
Name	Shop hanger
Color	White
Material	Polypropylene
Weight	60 g
Product thickness	1.2 mm

Table 3. Mold requirements

Item	Requirement
Molding type	Two-plate mold
Gate	Edge gate
Cavity and core	2
Mold base	Futaba

Table 4. Polypropylene properties

Property	Value	Unit
Density	0.90–0.91	g/cm³
Mold temperature	30–60	°C
Melt temperature	191–232	°C
Injection pressure	20–90	MPa

3.2 Filling Time Simulation Procedure

The filling time simulation was conducted in Autodesk Moldflow Adviser 2023 through a structured workflow. The product model was first imported into the software, followed by material selection, process-parameter definition, and execution of the Fill Time analysis. The main simulation setup steps are summarized in Figure 2. This workflow follows previous CAE-based injection-molding studies that used simulation to predict filling behavior and optimize mold/process conditions [11].

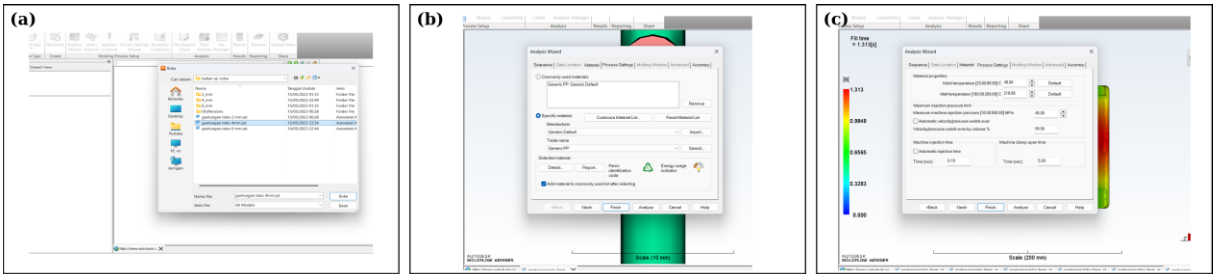

Figure 2. Simulation setup procedure in Autodesk Moldflow Adviser 2023: (a) importing the product model, (b) selecting the polypropylene material, and (c) setting the process parameters.

3.3 Best Gate Location

The selected gate location provided a more uniform distribution of melt flow, temperature, and pressure. Appropriate gate placement is important because it affects the flow-front pattern, cavity filling, and potential defect formation in injection molding [12]. The gate-location result and the corresponding filling-time distribution are shown in Figure 3.

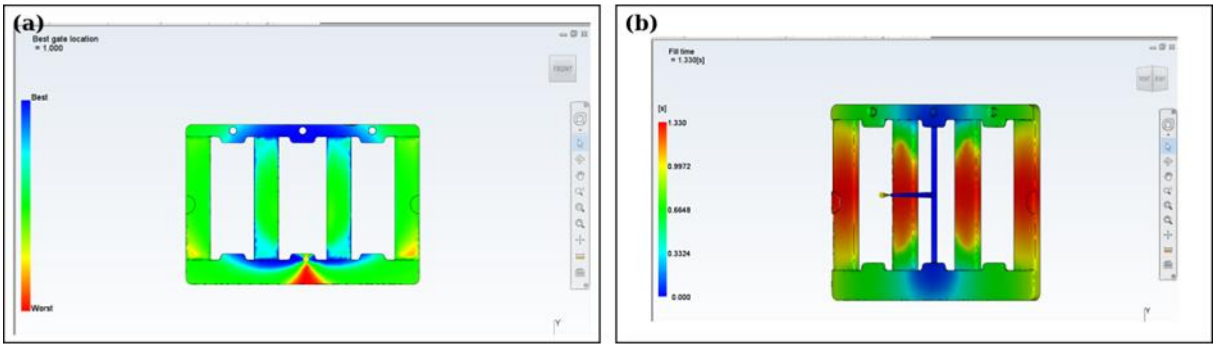

Figure 3. Gate-layout evaluation: (a) best gate location and (b) filling time for the selected gate layout.

Table 5. Initial simulation parameters

Parameter	Value
Sprue size	Initial diameter 4 mm
Sprue size	Final diameter 2 mm
Sprue length	50 mm
Gate size	Initial diameter 2 mm
Gate size	Final diameter 1 mm
Gate type	Edge gate
Runner size	4 mm
Runner type	Circular
Melt temperature	210 °C
Mold temperature	40 °C
Injection pressure	40 MPa

The analysis using the parameters in Table 5 showed that the maximum time required to fill the cavity was 1.32 s. Therefore, the gate placement was considered appropriate.

3.4 Runner Diameter Selection

Runner diameter was varied at 2, 4, and 6 mm using constant mold temperature, melt temperature, and injection pressure values as shown in Table 6.

Table 6. Runner variation parameters

Parameter	Value
Runner size	2, 4, and 6 mm
Mold temperature	40 °C
Injection pressure	40 MPa
Melt temperature	210 °C

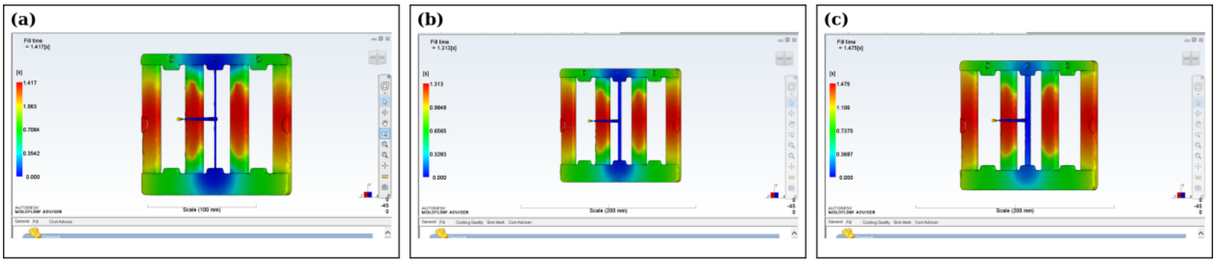

Figure 4. Filling time distributions for different runner diameters: (a) 2 mm, (b) 4 mm, and (c) 6 mm.

The runner-diameter comparison is shown in Figure 4. The 2 mm runner produced a filling time of 1.41 s, indicating that the smaller flow channel restricted molten plastic flow into the cavity. The 4 mm runner reduced the filling time to 1.31 s, whereas the 6 mm runner increased the filling time to 1.47 s. Therefore, the 4 mm runner was selected as the most suitable runner diameter because it produced the shortest filling time among the tested variations. This finding is consistent with previous studies showing that runner size affects flow resistance and filling-time performance [13], [14].

3.5 Pressure Variation Simulation

Injection pressure variation affected product filling. In this test, melt temperature and mold temperature were kept constant, while injection pressure was varied from 20 to 50 MPa.

Table 7. Pressure variation parameters

Parameter	Value
Melt temperature	210 °C
Mold temperature	40 °C
Injection pressure	20, 30, 40, and 50 MPa
Runner diameter	4 mm

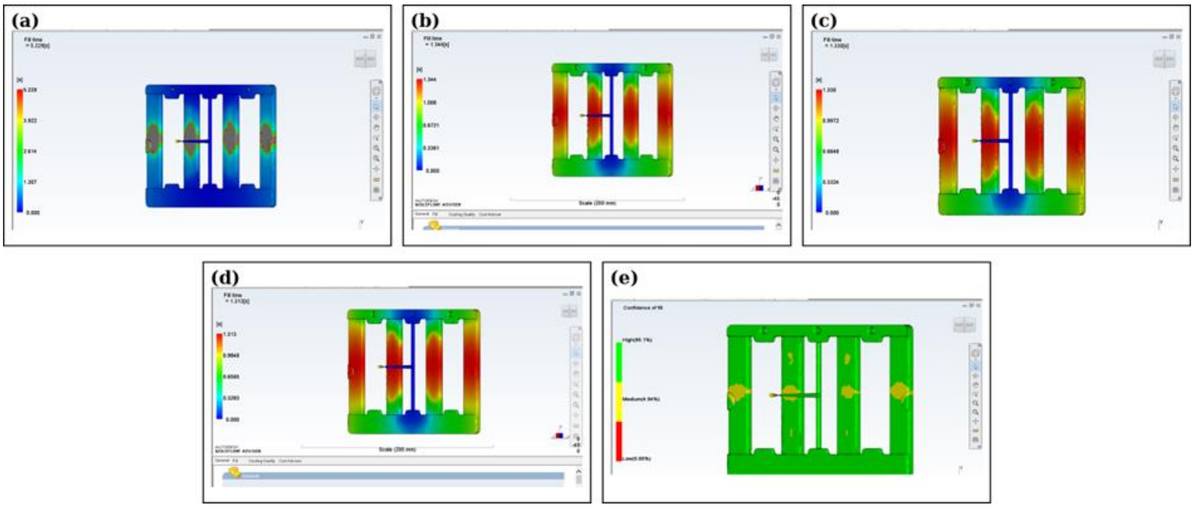

Figure 5. Filling time distributions at different injection pressures: (a) 20 MPa, (b) 30 MPa, (c) 40 MPa, (d) 50 MPa, and (e) confidence of fill.

Table 8. Pressure simulation results

Pressure (MPa)	Filling time (s)
20	5.20
30	1.334
40	1.313
50	1.313

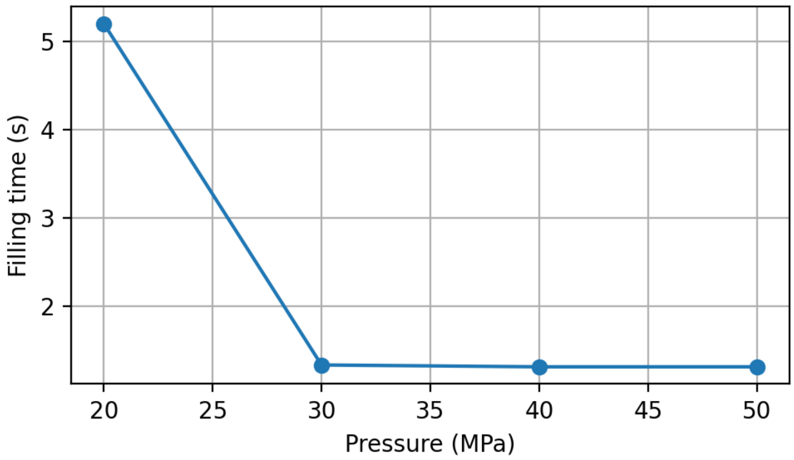

Figure 6. Graph of filling time results for pressure variation.

The effect of injection pressure on filling time is presented in Figure 5 and Table 8. At 20 MPa, the filling time was 5.20 s, and incomplete filling occurred because the injection pressure was insufficient to drive the molten polypropylene through the cavity. Increasing the pressure to 30 MPa reduced the filling time to 1.334 s. At 40 and 50 MPa, the filling time reached 1.313 s, indicating that further pressure increase produced no additional reduction under the tested conditions. The confidence-of-fill result also indicated that the selected parameter range was feasible. These findings confirm that injection pressure significantly affects filling behavior, as also reported in previous studies on injection-molding process parameters and pressure-related defects.

3.6 Temperature Variation Simulation

The effect of temperature was investigated by varying melt temperature at 180, 190, 200, 210, and 220 °C with constant mold temperature, pressure, and runner diameter.

Table 9. Temperature variation parameters

Parameter	Value
Melt temperature	180, 190, 200, 210, and 220 °C
Mold temperature	40 °C
Injection pressure	40 MPa
Runner diameter	4 mm

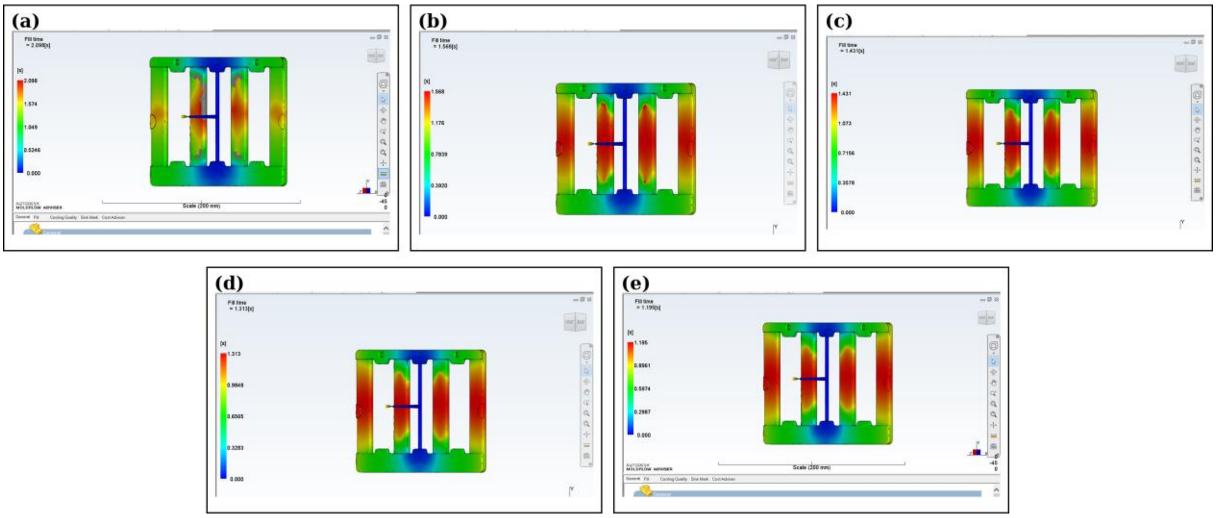

Figure 7. Filling time distributions at different melt temperatures: (a) 180 °C, (b) 190 °C, (c) 200 °C, (d) 210 °C, and (e) 220 °C.

Table 10. Temperature simulation results

Temperature (°C)	Filling time (s)
180	2.09
190	1.56
200	1.43
210	1.31
220	1.19

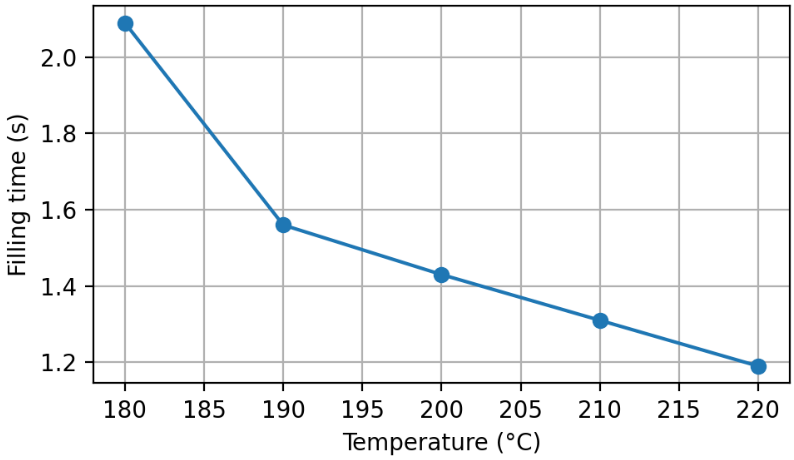

Figure 8. Graph of filling time results for temperature variation.

The effect of melt temperature on filling time is shown in Figure 7 and Table 10. At 180 °C and 40 MPa, the filling time was 2.09 s, and a short-shot defect was detected because the plastic melt did not flow sufficiently to fill the cavity. Increasing the melt temperature reduced the filling time to 1.56 s at 190 °C, 1.43 s at 200 °C, 1.31 s at 210 °C, and 1.19 s at 220 °C. This trend indicates that higher melt temperature reduced viscosity and improved flowability within the investigated range. However, an excessively high melt temperature may increase the risk of thermal degradation, discoloration, or burn marks. The observed trend supports previous studies showing that temperature variation affects filling time, shrinkage, and warpage behavior in injection molding.

3.7 Mold Design Result

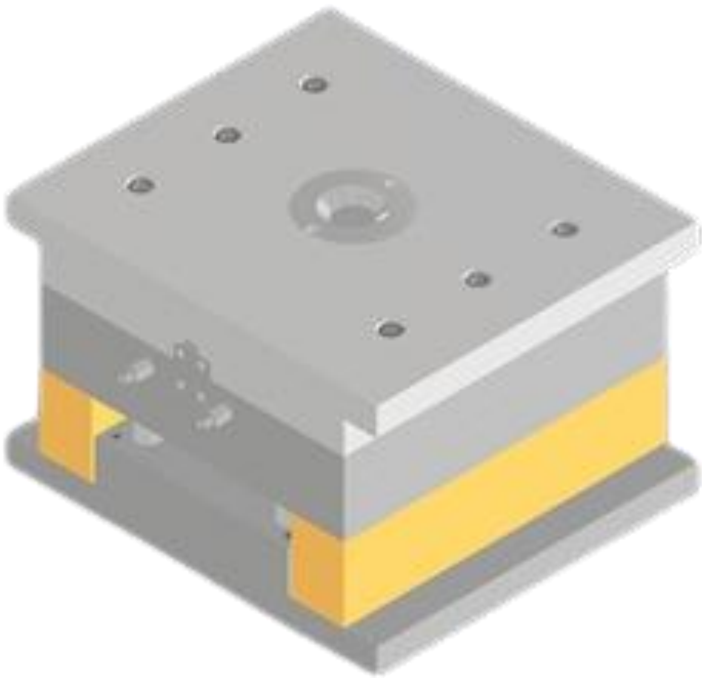

Figure 9. Three-dimensional mold design result.

The mold construction was designed based on a two-plate mold concept. The core and cavity size was 186 × 260 mm. The mold base used the Futaba SA series with a size of 330 × 350 mm and S55C material. The use of a suitable mold base, gate configuration, and clamping capacity is consistent with general injection-mold design considerations. The design required a clamping force of 470 kN. The machine available at CV. Mulus Teknik, a 90-ton UN90SKII injection machine with a clamping force of 900 kN, was suitable because its

3.8 Defects Observed During Simulation

Several defects were observed during the simulation, including short shot, weld lines, sink marks, and air traps. Representative simulation results are presented in Figure 10.

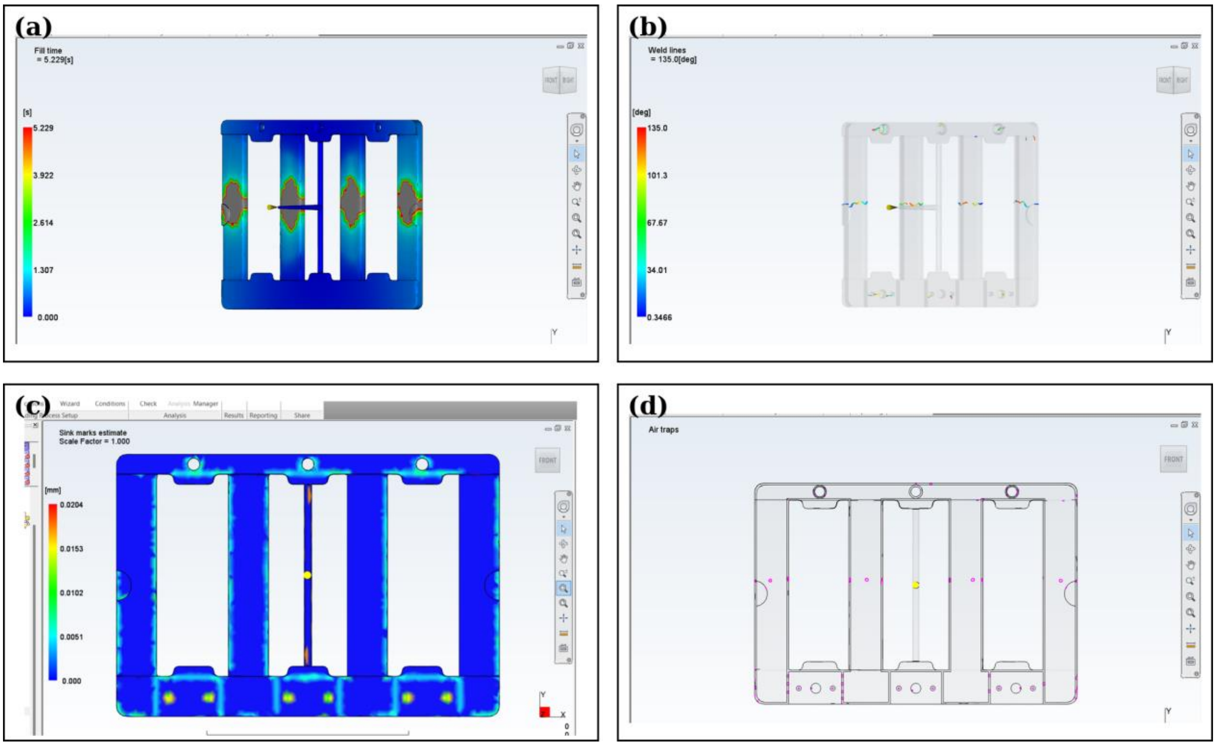

Figure 10. Defects observed during the simulation: (a) short shot, (b) weld lines, (c) sink marks, and (d) air traps.

A short shot occurs when molten plastic does not completely fill the mold cavity, generally because the injection pressure or melt temperature is insufficient. Weld lines are formed when two flow fronts meet and may reduce product strength. Sink marks appear as surface depressions and may be caused by nonuniform thickness or insufficient cooling. Air traps occur when air or gas is trapped in the cavity because it cannot escape properly during filling. Similar defects, including short shot, weld lines, sink marks, air traps, shrinkage, and warpage, have also been reported in previous injection-molding studies when process parameters and mold design were not properly optimized [4], [13].

4 Conclusion

Temperature and pressure affected the filling time of the shop hanger injection molding process. Lower temperature and pressure produced longer filling time, while higher temperature and pressure within the tested parameter range produced shorter filling time. The 4 mm runner diameter provided the optimal filling result among the tested runner sizes. The shortest filling time was obtained at a melt temperature of 220 °C, injection pressure of 40 MPa, and runner diameter of 4 mm. Defects detected during the simulation included weld lines, air traps, sink marks, and short shots. The short-shot defect occurred at 20 MPa and 210 °C because the pressure was insufficient to push the plastic material completely into the cavity. Weld lines and air traps occurred in the simulation because of flow-front convergence, while sink marks were mainly found around the runner area. The mold design used a core and cavity size of 186 × 260 mm and a Futaba SA series mold base of 330 × 350 mm with S55C material. The required clamping force was 470 kN, and the available 90-ton UN90SKII injection machine with a clamping force of 900 kN was suitable for production.

Acknowledgements

The authors would like to thank CV. Mulus Teknik for providing the production case study and technical information used in this research.

Conflicts of Interest

The authors declare no conflict of interest.

References

[1] I. Marwadi and H. Lubis, *Proses Manufaktur Plastik & Komposit*. Yogyakarta, Indonesia: Penerbit Andi, 2018.

[2] U. Wahyudi, “Pengaruh Injection Time dan Backpressure terhadap Cacat Injection Molding Menggunakan Material Polystyrene,” *Jurnal Teknik Mesin*, vol. 4, no. 3, pp. 81–90, 2015.

[3] S. Kamaruddin, A. Z. Khan, and S. H. Foong, “Application of Taguchi Method in the Optimization of Injection Moulding Parameters for Manufacturing Products from Plastic Blend,” *IACSIT International Journal of Engineering and Technology*, vol. 2, no. 6, pp. 574–580, 2010.

[4] T. Rusianto, S. A. Ellyawan, and A. Rahmanto, “Shrinkage pada Plastik Bushing dengan Variabel Temperatur Injeksi Plastik,” *Jurnal Kompetensi Teknik*, vol. 2, no. 1, pp. 65–72, 2010.

[5] Sugondo, “Minimalisasi Cacat dengan Pengaruh Tekanan terhadap Kualitas Produk pada Proses Injection Molding dengan Menggunakan Simulasi,” *Jurnal Teknik Mesin*, pp. 30–40, 2007.

[6] D. Zulianto, “Analisa Pengaruh Variasi Suhu Plastik terhadap Cacat Warpage dari Produk Injection Molding Berbahan Polypropylene (PP),” Universitas Muhammadiyah Surakarta, 2015.

[7] G. Enggartyasto, “Simulasi dan Analisis Injeksi Plastik untuk Produk Kelistrikan Berbasis Software CAE (Autodesk Moldflow Adviser) dengan Mold Design,” *Jurnal Teknik Mesin Universitas Islam Indonesia*, vol. 1, pp. 56–90, 2022.

[8] A. Erfina, “Optimasi Desain Mold untuk Mereduksi Cacat Flash dan Shrinkage pada Produk Paku Kotak dengan Menggunakan Software Simulasi Moldflow,” *Jurnal Manufaktur*, vol. 8, no. 1, 2015.

[9] S. A. Riyanto, P. W. Anggoro, and C. Budiantoro, “Optimalisasi Proses Injeksi Molding Menggunakan Moldflow Dual-Domain pada Desain Base Plate,” *Jurnal Jurusan Teknik Industri, Universitas Atma Jaya Yogyakarta*, pp. 71–80, 2015.

[10] R. A. Zuliansyah, “Simulasi dan Analisis Injeksi Plastik untuk Produk-Produk Kelistrikan Berbasis Software CAE (Autodesk Moldflow),” 2023.

[11] R. Rudiyadi, “Optimasi Fill Time Injection Molding Crisper dengan Bantuan Software Autodesk Moldflow,” *Jurnal Teknik Mesin*, vol. 15, no. 1, 2016.

[12] I. Yulianto, Rispianda, and H. Prassetiyo, “Rancangan Desain Mold Produk Knob Regulator Gas pada Proses Injection Molding,” *Reka Integra*, vol. 2, no. 3, pp. 140–151, 2014.

[13] E. K. Laksanawati, S. Lestari, and S. Pramono, “Analisa Variasi Tekanan, Temperature, Ukuran Runner terhadap Filling Time Optimal pada Injeksi Molding,” *Jurnal Universitas Muhammadiyah Tangerang*, vol. 3, no. 1, pp. 2–8, 2021.

[14] M. Rizal, “Pengaruh Variasi Tekanan, Temperatur, dan Ukuran Runner terhadap Filling Time pada Proses Injeksi Molding Produk Penghapus Whiteboard,” *Jurnal Teknik Mesin Universitas Islam Indonesia*, 2018.