Otopro Volume 21 No. 2 May 2026 pages. 78-84
P-ISSN: 1858-411x; E-ISSN: 2685-7863
DOI: 10.26740/otopro.v21n2.p78-84 https://journal.unesa.ac.id/index.php/jo
(This journal accommodates review, experimental, and thinking related to mechanical engineering and its application)

EVALUATION OF INJECTION MOLDING PROCESS PARAMETERS ON THE QUALITY OF BRAKE FLUID RESERVOIR CAP PRODUCTS USING MOLDFLOW SIMULATION

Adinda Rahmah Shalihah[1*], *Aditya* Bagja Subarkah[1], *Farras* Novyansyah[1], *Irfan* Fadillah[1],
Ammar Fathi Muhadzab[1], *Hafizh* Maulana Alfath[1], *Ulima* Adristi Azis[1], and *Hikari* Qurrata'ain Nurhadi[1,2]

[1]Automotive Engineering Technology, Polytechnic STMI Jakarta, Jakarta, 10510, Indonesia
[1]adindarahmah@stmi.ac.id
[2]Directorate General of Metal, Machinery, Transportation Equipment, and Electronics Industry, Ministry of Industry, Jakarta, 12950, Indonesia
[2]hikariqurrata@kemenperin.go.id

Abstract. This study aims to analyze the effect of melt temperature and mold surface temperature variations on the quality of the motorcycle brake fluid reservoir cap using Autodesk Fusion simulation. Three parameter variations were applied: 260°C/80°C, 220°C/50°C, and 180°C/20°C, while other process parameters were kept constant. The evaluation focused on fill confidence, weld lines, sink marks, and warpage. The results show that all conditions achieved 100% fill confidence with a filling time of 0.53 seconds. However, higher temperatures reduced weld lines but increased sink marks and warpage. Conversely, lower temperatures minimized deformation but increased the number of visual defects. The medium temperature condition provided the best balance between visual quality and dimensional stability. Therefore, the 220°C/50°C condition is recommended as optimal.

Keywords: injection molding; moldflow simulation; product defects.

1 Introduction

The braking system in motor vehicles is a vital safety feature that demands exceptional material dependability and precise dimensional accuracy. One of the key components in this system is the brake fluid reservoir cap (master cylinder cap), which maintains hydraulic pressure and prevents contamination of the brake fluid by external contaminants. Given its crucial role, the manufacturing process using injection molding must be strictly controlled to ensure high product quality [1].

Injection molding is widely utilized in manufacturing plastic components due to its ability to deliver high production speeds, low costs, and complex geometries with excellent dimensional accuracy. However, the quality of the final product is heavily influenced by both the process parameters and mold design, especially in relation to common defects such as warpage, shrinkage, sink marks, weld lines, and short shots [2], [3].

As technology advances, numerical simulation tools such as Autodesk Fusion are being increasingly utilized to analyze material flow behavior and temperature distribution during injection molding processes. Through simulation, potential product defects can be predicted at an early stage and process parameters optimized before physical mold fabrication. Previous studies have shown that simulation-based approaches, including Moldflow analysis, are effective for evaluating filling behavior, shrinkage, and warpage, as well as reducing the need for trial-and-error in mold manufacturing [4]. As a result, the need for trial-and-error in manufacturing can be minimized, leading to improved efficiency in both time and cost. Furthermore, simulation enhances the design process by allowing virtual evaluation of mold performance prior to production [5], [6].

Early defect identification is a crucial preventive step in the manufacturing industry. According to recent studies, mold design errors or improper process parameters can lead to significant production losses if

---

[*] Corresponding author: adindarahmah@stmi.ac.id

not detected at an early stage [7]. To minimize the risk of failure, flow simulation is highly effective in optimizing process parameters and predicting defect-prone areas before physical mold fabrication is carried out [8].

However, studies specifically analyzing the effects of melt temperature variations on thermal conditions during component production remain limited. Therefore, this research focuses on analyzing the influence of melt temperature variations on the production results of motorcycle brake fluid reservoir caps through simulation using Autodesk Fusion. Based on simulation data at various temperature levels, this study aims to identify the most stable thermal conditions. The results of this study are expected to offer valuable insights for optimizing injection molding process parameters, ultimately enhancing both the quality and safety of automotive components.

2 Method

This study utilizes an experimental analysis based on the modeling of the injection molding process to explore how variations in parameters impact the quality of the motorcycle brake fluid reservoir cap components. This approach is carried out by varying the melt temperature and mold surface temperature, which are then analyzed to determine the resulting product quality. Temperature parameters play a crucial role in determining product quality, as they significantly affect material flow, viscosity, and the likelihood of defect formation [9]. Figure 1 illustrates the overall research workflow.

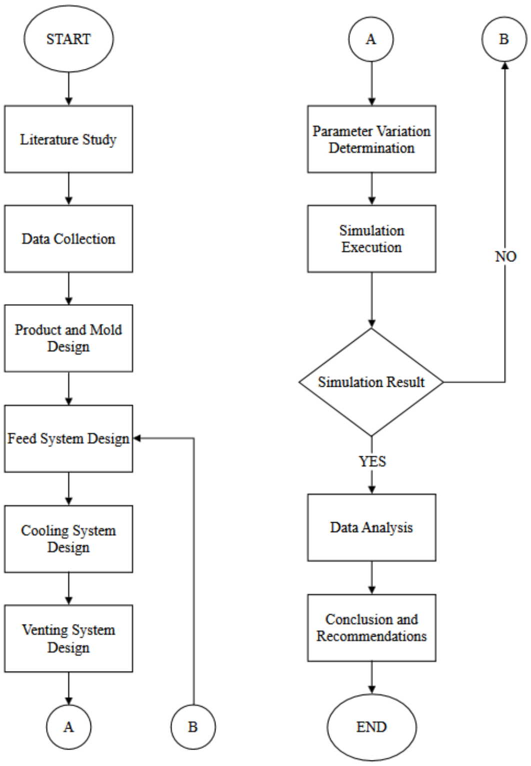


Fig. 1. Flowchart

2.1 Product and Mold Design

The product developed in this study is a motorcycle brake fluid reservoir cap, which functions to maintain hydraulic pressure and prevent contamination of the brake fluid from the external environment. The component includes specific features on its lower section that help maintain its position and prevent displacement when installed on the brake fluid reservoir. Additionally, a small hole is located at the center of the component to allow trapped air to escape when the reservoir is sealed. Figure 1 illustrates the design of the motorcycle brake fluid reservoir cap.

The mold design was developed based on the Handbook LKM and Catalog 2530 [10], which serve as primary references for mold fabrication of the brake reservoir cap component. Improper design parameters—such as gate, sprue, runner, and venting—can lead to various product defects, including shrinkage, sink marks, weld lines, warpage, short shots, and poor surface quality [11].

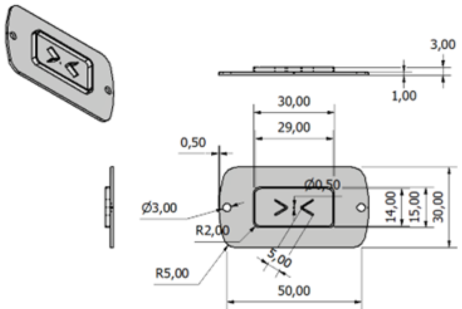


Fig. 2. *Brake Fluid Reservoir Cap Products*

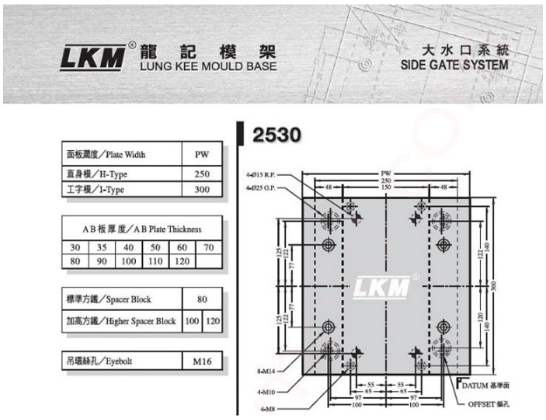


Fig. 3. Mold Design

2.2 Feed System Design

The feeding system is designed to ensure that molten material flows evenly from the nozzle into the cavity. The primary components of the mold include the sprue, runner, gate, and cavity. The sprue acts as the channel connecting the nozzle to the runner. The molten plastic enters the mold through the sprue, flows through the runner system, and then enters the cavity through the gate [12].

In this study, the sprue was not designed through a custom geometric approach but instead utilized a standard component from MISUMI. The selection was made based on the material flow requirements and its compatibility with the injection molding machine nozzle. Using standardized components aims to simplify the manufacturing process while ensuring consistent dimensions in line with industry standards.

The feed system was calculated to determine the optimal dimensions of the sprue, runner, and gate to ensure stable material flow without obstruction [13]. The sprue size used follows the MISUMI standard with a diameter of 3.5 mm. Meanwhile, the runner diameter is determined using the following equation:

$$D = S_{\max} + 1.5\ mm \tag{1}$$
$$D = 3\ mm + 1.5\ mm = 4.5\ mm$$

Where:
D = runner diameter (mm)
$S_{\max}$ = maximum product thickness (mm)
The gate type used is an edge gate with a rectangular shape. The depth and width dimensions of the gate are required to define this gate type. The gate thickness is calculated using the following equation:

$$t = n \times s \tag{2}$$
$$t = 0.7 \times 1\ mm = 0.7\ mm$$

Where:
t = gate thickness (mm)
s = product thickness (mm)
n = material constant

The material constant for polypropylene is 0.7. The width of the edge gate is calculated using the following equation:

$$w = (n \times \sqrt{A}) / 30 \tag{3}$$
$$w = (0.7 \times 1593) / 30 = 0.93\ mm$$

Where:
w = gate width (mm)
A = product area (mm^2) = 1,593 mm^2

Based on the calculations above, the edge gate thickness is 0.7 mm, while the gate width is 0.93 mm. Figure 4 presents the cavity layout design of the brake fluid reservoir cap component.

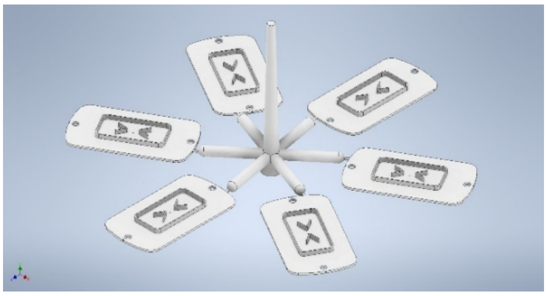


Fig. 4. Layout Cavity

2.3 Cooling System Design

The cooling system is designed to control heat dissipation from the material after the cavity filling process. Cooling channels are arranged around the cavity at specific distances to ensure uniform cooling distribution [14].

The cooling system is a critical part of the injection molding process, as it accounts for approximately 60-80% of the total cycle time. Therefore, an optimal cooling system design is required to achieve more uniform temperature distribution and to reduce the potential for product defects such as warpage and shrinkage [15].

The design of the coolant system aims to regulate the temperature distribution during the cooling phase. The distance between the coolant channel and the surface of the cavity or core is calculated using the following equation:

$$S = (2{\sim}3) \times D \tag{4}$$
$$S = 3 \times 6 = 18\ mm$$

Where:
S = distance between the coolant channel and the cavity or core surface (mm)
D = coolant channel diameter (mm)

2.4 Venting System Design

The venting system is designed to expel trapped air or gases from the mold cavity to prevent the occurrence of air traps and other related defects. Based on the characteristics of polypropylene (PP), which has low viscosity, the vent depth is set at 0.015 mm. This dimension is carefully selected to allow air to escape efficiently while preventing molten plastic from flashing out.

The venting design uses a stepped configuration (relief venting) with a 1 mm width. The initial section of the vent channel (land) is designed with a depth of 0.015 mm and extends 2 mm from the edge of the cavity [16], which is then connected to a secondary exhaust channel with a depth of 0.5 mm leading to the external environment. This stepped design provides an unobstructed pathway for gas evacuation without compromising the mold plate's structural integrity [17]. Figure 5 illustrates the overall mold design, and Table 1 lists the mold components.

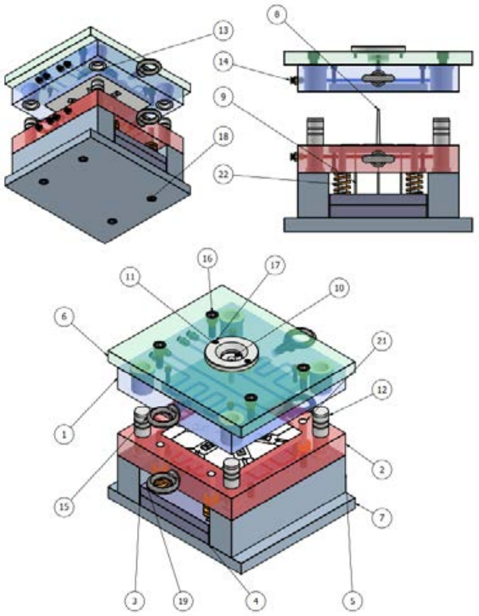


Fig. 5. Mold Assembly

Table 1. Mold Component List

No	Name	Material	Qty
1	Cavity Plate	AISI P20	1
2	Core Plate	AISI P20	1
3	Ejector Plate	AISI P20	1
4	Retainer Plate	AISI P20	1
5	Spacer	AISI P20	2
6	Top Plate	AISI P20	1
7	Bottom Plate	AISI P20	1
8	Product	PP	1
9	Ejector Pin	MISUMI	12
10	Sprue Bush	MISUMI	1
11	Locating Ring	MISUMI	1
12	Guide Pin	MISUMI	4
13	Guide Bush	MISUMI	4
14	Quick Fitting	MISUMI	8
15	Lifting Eyebolt	MISUMI	4
16	M14 x 35	SS304	4
17	M6 x 16	SS304	2
18	M14 x 120	SS304	4
19	M10 x 35	SS304	4
20	M8 x 30	SS304	8
21	Return Pin	MISUMI	4
22	Spring	MISUMI	4
23	Core Insert	AISI P20	1
24	Cavity Insert	AISI P20	1

2.5 Parameter Variation

The testing was conducted using three variations of melt temperature and mold temperature, namely High, Medium, and Low, as presented in Table 2. Each variation was tested while keeping the other process parameters constant.

The variations in each parameter were compared to assess the impact of temperature on the formation of defects such as air traps, weld lines, sink marks, and warpage. The evaluation was carried out based on the magnitude of shrinkage and the location of defects on the product.

Table 2. Simulation Parameters

Condition	Melt Temperature	Mold Temperature
High	260°C	80°C
Medium	220°C	50°C
Low	180°C	20°C

3 Results and Discussion

3.1 Flow Distribution Analysis

After the machine setup parameters were determined, the next stage involved analyzing the flow distribution. This step is conducted to assess the material's ability to fill the mold cavity and observe the flow behavior throughout the injection process, along with the occurrence of warpage. The analysis is conducted to verify whether the designed feed system can uniformly distribute the molten material across all cavities.

Figure 6 presents the flow distribution results, showing that the material flows smoothly and uniformly, requiring approximately 0.53 seconds to completely fill the entire cavity. This relatively short filling time indicates that the selected processing parameters and feed system design are effective in promoting efficient material flow.

Achieving consistent flow distribution in injection molding is crucial for reducing the likelihood of defects. The absence of significant flow imbalances suggests that the runner and gate design are well-optimized, enabling consistent filling behavior across the cavity. Furthermore, the smooth flow pattern indicates stable processing conditions, which contribute to improved product quality and reduced likelihood of structural defects.

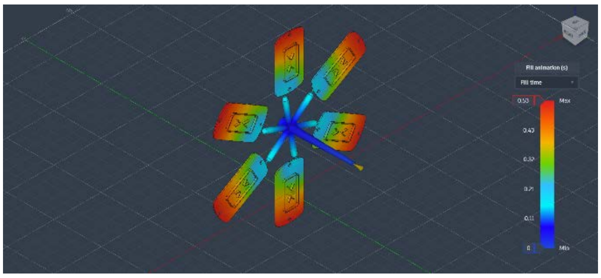


Fig. 6. Flow Distribution (Medium Condition)

3.2 Air Trap Defect Analysis

This analysis was conducted to evaluate the venting system's effectiveness and determine whether the material flow during injection allows proper air evacuation from the cavity. By examining the presence and distribution of air traps, the study aims to ensure that the mold design and processing parameters support defect-free filling conditions.

Based on results across all three parameter variations (High, Medium, and Low), no air-trap defects were detected. This indicates that the venting system design is effective and that the molten material flow provides sufficient air release during injection. The absence of air traps also suggests that the combination of gate, runner, and venting design provides balanced flow, thereby reducing the likelihood of gas entrapment and improving product quality.

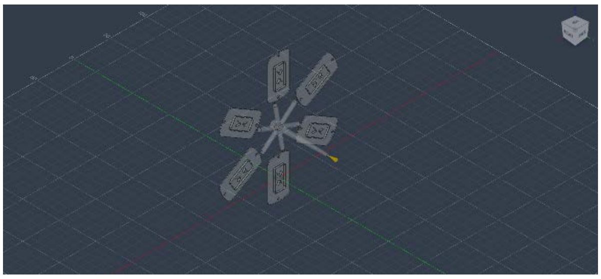
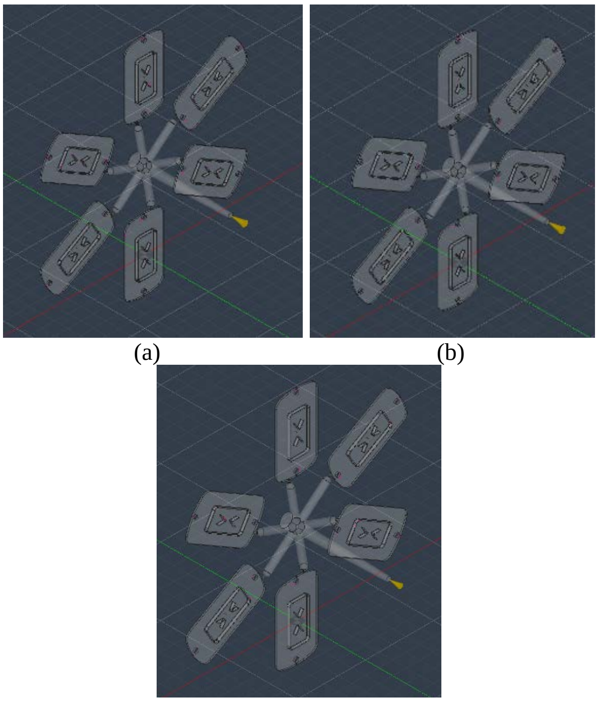


Fig. 7. Air Trap Defect (Medium)

3.3 Weld Line Defect Analysis

This analysis was conducted to assess the impact of process temperature on the formation of weld lines and to determine how variations in melt and mold temperatures influence the material’s ability to properly fuse at the flow junctions. Understanding this behavior is essential for improving product strength and minimizing structural defects.



(a) (b)

(c)

Fig. 8. Weld Line Defect, (a) Low ; (b) Medium ; (c) High

The results show that the number of weld lines increases as the process temperature decreases:

- High Condition (260°C / 80°C): 33 weld lines
- Medium Condition (220°C / 50°C): 38 weld lines
- Low Condition (180°C / 20°C): 45 weld lines

This trend suggests that higher temperatures promote better flowability and improve the fusion of molten material at flow fronts, thereby reducing the formation of weld lines. Conversely, at lower temperatures, the material viscosity increases, leading to poorer bonding between flow fronts and more weld lines.

While it is not possible to completely avoid weld lines in complex geometries, their effects can be mitigated by fine-tuning key process parameters, such as melt temperature, mold temperature, and injection pressure, along with refining the design of the gate and runner. In conclusion, the findings emphasize the critical role of controlling processing temperatures to improve the overall integrity of the product and minimize the structural flaws linked to weld line formation.

3.4 Sink Mark Defect Analysis

This analysis was conducted to evaluate the presence and severity of sink mark defects under different processing temperature conditions, as well as to understand how melt temperature and mold temperature influence material shrinkage behavior during cooling.

The results show that sink mark defects were identified in all parameter variations, with the same number of affected areas (six locations). However, the depth of the sink marks varies slightly across different conditions:

- High Condition (260°C / 80°C): 0.008 mm
- Medium Condition (220°C / 50°C): 0.007 mm
- Low Condition (180°C / 20°C): 0.007 mm

These findings indicate that although temperature variations do not significantly affect the number of sink mark locations, they have a minor influence on the depth of the defects. The slightly higher sink mark depth at elevated temperatures may be attributed to increased material shrinkage due to prolonged cooling time.

To reduce sink mark defects, preventive actions should focus on enhancing the cooling system for even heat distribution, adjusting the holding pressure and packing time to account for material shrinkage, and refining the product design to ensure uniform wall thickness. Moreover, optimizing gate placement and improving material flow throughout the mold can further minimize the chances of localized shrinkage.

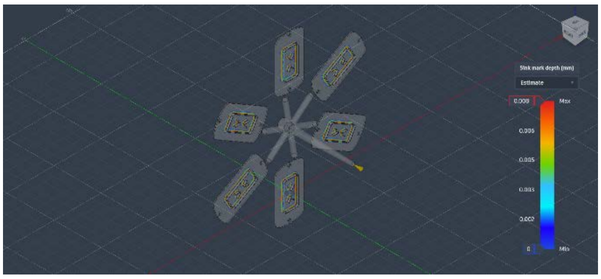


Fig. 9. Sink Mark Defect (High)

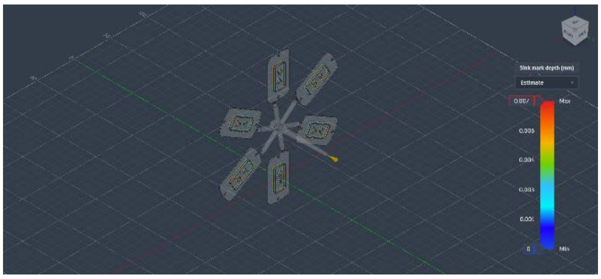


Fig. 10. Sink Mark Defect (Medium)

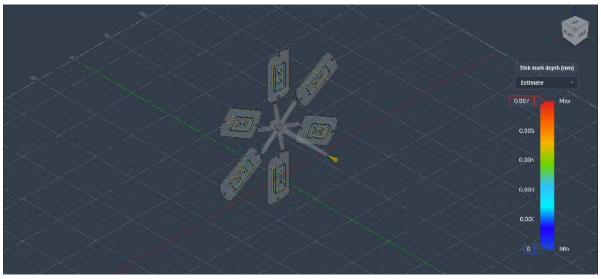


Fig. 11. Sink Mark Defect (Low)

3.5 Warpage Defect Analysis

This analysis was performed to assess how variations in process temperatures affect warpage and to explore how changes in melt and mold temperatures influence thermal gradients and shrinkage behavior in the molded component.

The simulation results show that as the processing temperature decreases, the warpage value also decreases:

- High Condition (260°C / 80°C): 1.976 mm
- Medium Condition (220°C / 50°C): 1.804 mm
- Low Condition (180°C / 20°C): 1.576 mm

This trend suggests that higher processing temperatures tend to increase thermal gradients and residual stresses within the material, leading to greater deformation after cooling. Conversely, lower temperatures reduce the degree of shrinkage imbalance, resulting in improved dimensional stability and lower warpage values.

To reduce warpage, various approaches can be employed, such as fine-tuning the cooling system for even temperature distribution, modifying key processing parameters like melt temperature, mold temperature, and injection pressure, and ensuring even material flow by optimizing the gate and runner design. Moreover, maintaining a consistent wall thickness and applying the right packing pressure can help minimize internal stresses, further preventing warpage.

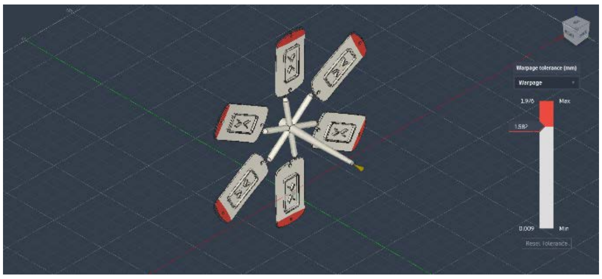


Fig. 12. Warpage Defect (High)

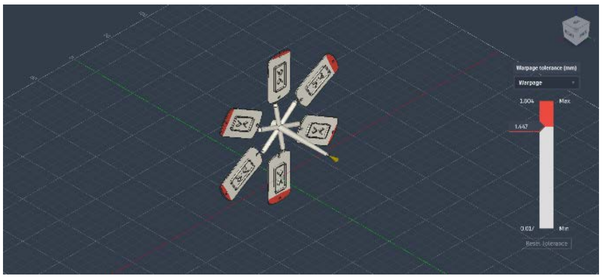


Fig. 13. Warpage Defect (Medium)

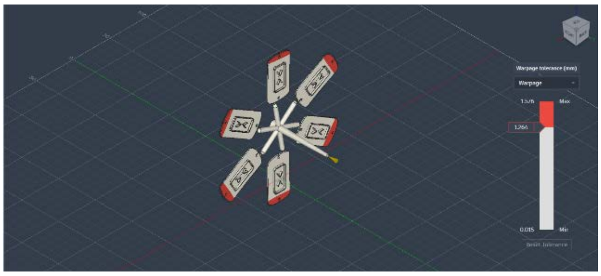


Fig. 14. Warpage Defect (Low)

4 Conclusion

Based on the simulation results, variations in melt temperature and mold surface temperature significantly affect defect formation in the motorcycle brake fluid reservoir cap. No air-trap defects were found across all three parameter variations, indicating that the venting system and flow design were effective at releasing trapped air during the injection process. However, weld lines, sink marks, and warpage were still detected with different levels of severity. The number of weld lines increased as the process temperature decreased, from approximately 33 lines at the High parameter to 45 lines at the Low parameter. Meanwhile, sink marks appeared in the same number of areas for all parameters, with only a slight difference in depth. Warpage showed a decreasing trend as the temperature was reduced, with the lowest value obtained at the Low parameter. For clearer simulation results, the detailed data are presented in Table 3.

Table 3. Simulation Result

Defects	**Condition**		
	High	**Medium**	**Low**
Air Trap	0	0	0
Weld Line	33 faces	38 faces	45 faces
Sink Mark	6 faces 0.008 mm	6 faces 0.007 mm	6 faces 0.007 mm
Warpage	1.976 mm	1.804 mm	1.576 mm

In conclusion, the Medium parameter, with a melt temperature of 220°C and a mold surface temperature of 50°C, yields the most balanced results. This setting results in a relatively low warpage value of 1.804 mm, minimal sink mark depth of 0.007 mm, and a moderate number of weld lines compared to the High and Low parameters. While the Low parameter produces the least warpage, it also leads to a higher number of weld lines, which could compromise the product's mechanical strength. Therefore, the Medium parameter is deemed the most optimal, offering the best trade-off between dimensional stability, surface finish, and structural integrity of the molded product.

References

[1] R. Farooque, M. Asjad, and S. J. A. Rizvi, "A current state of art applied to injection moulding manufacturing process – A review," *Mater. Today Proc.*, vol. 43, pp. 441–446, 2021, doi: https://doi.org/10.1016/j.matpr.2020.11.967.

[2] N. Zhao, J. Lian, P. Wang, and Z. Xu, "Recent progress in minimizing the warpage and shrinkage deformations by the optimization of process parameters in plastic injection molding: a review," *Int. J. Adv. Manuf. Technol.*, vol. 120, no. 1, pp. 85–101, 2022, doi: 10.1007/s00170-022-08859-0.

[3] A. Mourya, A. Nanda, K. Parashar, Sushant, and R. Kumar, "An explanatory study on defects in plastic molding parts caused by machine parameters in injection molding process," *Mater. Today Proc.*, vol. 78, pp. 656–661, 2023, doi: https://doi.org/10.1016/j.matpr.2022.12.070.

[4] J. Chen, Y. Cui, Y. Liu, and J. Cui, "Design and Parametric Optimization of the Injection Molding Process Using Statistical Analysis and Numerical Simulation," 2023. doi: 10.3390/pr11020414.

[5] M. Baum and D. Anders, "A numerical simulation study of mold filling in the injection molding process," *Comput. Methods Mater. Sci.*, vol. 21, Sep. 2021, doi: 10.7494/cmms.2021.1.0743.

[6] M. Baum, D. Anders, and T. Reinicke, "Approaches for Numerical Modeling and Simulation of the Filling Phase in Injection Molding: A Review," 2023. doi: 10.3390/polym15214220.

[7] M. Kariminejad, D. Tormey, C. Ryan, C. O'Hara, A. Weinert, and M. McAfee, "Single and multi-objective real-time optimisation of an industrial injection moulding process via a Bayesian adaptive design of experiment approach," *Sci. Rep.*, vol. 14, no. 1, p. 29799, 2024, doi: 10.1038/s41598-024-80405-2.

[8] K. T. Chiwapon Nitnara, "Simulation-Based Optimization of Injection Molding Process Parameters for Minimizing Warpage by ANN and GA," *Int. J. Technol.*, vol. 14, no. 2, pp. 291–319, 2023, doi: https://doi.org/10.14716/ijtech.v14i2.5573.

[9] J. Jeon, B. Rhee, and J. Gim, "Melt Temperature Estimation by Machine Learning Model Based on Energy Flow in Injection Molding," 2022. doi: 10.3390/polym14245548.

[10] L.K.Group, "LKM Lung Kee Mould Base Handbook," 2020.

[11] A. Kaswadi and I. Tauhid, "Optimalisasi Perancangan Runner dan Gate Cetakan Injeksi Plastik dengan Metode Simulasi," pp. 15–20, 2017.

[12] P. D. Kale, P. D. Darade, and A. R. Sahu, "A literature review on injection moulding process based on runner system and process variables," *IOP Conf. Ser. Mater. Sci. Eng.*, vol. 1017, no. 1, p. 12031, 2021, doi: 10.1088/1757-899X/1017/1/012031.

[13] R. S. Theryo, *Teknologi Plastic Injection Mold (Cet. 1)*. Yogyakarta: PT.Kanisius, 2022.

[14] Y. Wang and C. Lee, "Design and Optimization of Conformal Cooling Channels for Increasing Cooling Efficiency in Injection Molding," 2023. doi: 10.3390/app13137437.

[15] J. Li, Y. C. Ong, and W. M. Wan Muhamad, "Optimization Design of Injection Mold Conformal Cooling Channel for Improving Cooling Rate," 2024. doi: 10.3390/pr12061232.

[16] V. Goodship, A. (Firm), and R. T. Limited, *Practical Guide to Injection Moulding*. Rapra Technology, 2004. [Online]. Available: https://books.google.com.sg/books?id=zFZT0acVel4C

[17] M. Krishnappa, S. Nagaraja, N. Munegowda, and M. I. Ammarullah, "Design and Analysis of Injection Molds for Plastic Components for Biomedical Applications," *Eng. Reports*, vol. 7, 2025, [Online]. Available: https://api.semanticscholar.org/CorpusID:277199410