JMPM: Jurnal Material dan Proses Manufaktur
Vol.8, No.2, p146-156, Desember 2024
DOI: https//doi.org/10.18196/jmpm.v8i2.26118

Design and Simulation of a 12-Cavity Injection Mold with Optimized Runner System for Efficient Production of Sample Cups

Cahyo Budiyantoro[a], Wahyu Aji Pratama[b]
[a] Program Studi Program Profesi Insinyur Universitas Muhammadiyah Yogyakarta, Jl. Brawijaya, Geblagan, Tamantirto, Kec. Kasihan, Kabupaten Bantul, Daerah Istimewa Yogyakarta 55183
[b] Program Studi Teknik Mesin Universitas Muhammadiyah Yogyakarta, Jl. Brawijaya, Geblagan, Tamantirto, Kec. Kasihan, Kabupaten Bantul, Daerah Istimewa Yogyakarta 55183

Correspondence author: cahyo_budi@umy.ac.id

Keywords:

Injection molding; runner optimization; three mold plates; sample cups.

ABSTRACT

Injection molding is widely used in large-scale manufacturing due to its efficiency in producing plastic products. One such product is the sample cup, used in the medical field for liquid sample storage. An inefficient runner system in injection molding can cause uneven material distribution, leading to defects and material waste. This study aims to simulate the injection molding process for a 12-cavity mold, optimizing the runner system and designing a three-plate mold for sample cup production. Product and mold designs were created using Autodesk Inventor Professional, and injection molding simulations were performed using Moldflow Plastic Insight to determine the optimal runner layout. The simulation results indicated that runner layout 1 was superior, with improved performance in terms of fill time (0.8028 s), pressure at switchover (85.93), sink index (0.4359%), volumetric shrinkage (10.50%), and cycle time (50 s). Based on these results, layout 1 was selected for the three-plate mold.

1. INTRODUCTION

Plastics have become an integral part of daily life and are widely used across various industries, from food and beverage packaging to household items and medical equipment. The strength, light weight, flexibility, and relatively low cost of plastics make them preferred material for mass production processes. Consequently, plastic products are among the most used materials in everyday life. Considering the large number of plastic-based products produced globally, this research aims to design a plastic mold for manufacturing in Indonesia. The case study focuses on the design of a sample cup, a container used to store liquid samples for medical and laboratory testing. The sample cup, made from polypropylene, features a transparent design with an integrated lid and small container structure.

The use of plastic in medical products allows for cost-effective production through injection molding. The use of plastic materials in medical products also helps minimize contamination during diagnostic processes, and the plastic waste can be recycled with proper treatment [1]. However, traditional two-mold plate systems have the disadvantage of requiring manual separation of the runner from the product, leading to inefficiency. To address this, the three-mold plate system is applied in this study. A three-mold plate system is designed for complex multi-component plastic parts, consisting of three separate plates that allow the runner to be automatically detached from the product when the mold opens, improving production efficiency [2].

A common method for reducing manufacturing costs is increasing production capacity, achievable through multi-cavity molds. Multi-cavity molds are typically designed with a fishbone-type runner system. However, this runner system is prone to filling imbalance during injection. High shear areas near the mold walls can cause uneven temperature distribution, affecting material viscosity and leading to imbalances when the polymer enters the gate, resulting in product quality issues [3].

Short cycle time is crucial for high productivity, and minimizing cycle time without compromising product quality is essential. Defects such as deformation, volumetric shrinkage, and weld lines must be avoided to maintain product dimensions and appearance. Achieving an optimal injection molding cycle requires careful control of the filling, packing, and cooling phases. Process parameters like melt temperature, packing pressure, packing time, injection time, and cooling time must be optimized to reduce warpage and cycle time simultaneously [4].

Key design parameters for high-quality molds include the runner system type, cooling system, and three-mold plate design. To address these challenges, simulation-based design, accurate calculations, and planning are crucial [5]. Optimal mold design requires considering material properties, product shape, mold material, and machine features to ensure efficient production[6]. Simulation software plays a critical role in analyzing mold filling, ensuring that the mold cavities are filled optimally under specified injection pressure. This step helps identify the ideal pressure needed for optimal filling and ensures the mold is fully filled while minimizing potential molding errors [7].

The three-mold plate method in injection molding is widely applied in industry due to its optimized design calculations. One key factor is the use of a trapezoidal runner, which enhances efficiency by reducing material filling time [8]. The runner system, connecting the sprue and gate, plays a vital role in ensuring product quality. Key parameters such as runner design, diameter, and cavity layout influence performance. Runner balance analysis is essential for optimizing flow, reducing fill time, and minimizing defects [9].

Proper gate placement is crucial for maintaining product aesthetics and functionality. Strategically concealed gates improve appearance, while pin gates facilitate easy separation of the product from the runner and reduce shear stress [10][11]. To enhance efficiency, simulations using Autodesk Moldflow Plastic Insight help in optimizing runner balance and filling time before actual production [12].

Material selection is critical, especially for medical and chemical containers. Polypropylene (PP) is widely used due to its chemical resistance, low density (0.735 g/mL), and high melt temperature (168-200°C), making it suitable for various applications [13], [14], [15]. Cooling time significantly affects product quality, as improper settings can lead to warpage. Optimizing cooling time helps prevent defects and ensures efficiency.

Machine selection is equally important, particularly clamping force, which determines the pressure applied during the injection process. Simulation analysis aids in identifying the required clamping force for different molds. For example, the Jingqiong NHTX/M injection molding machine has a clamping force of 4.44 tons and a maximum locking force of 35.69 tons, making it suitable for various production needs [16].

2. DESIGN METHOD

2.1 Designed Product

A sample cup, as described in Figure 1, is a small container designed for holding liquid or solid samples, primarily used in laboratories and medical applications. It is commonly used for diagnostic testing, chemical analysis, and sample storage. Sample cups are typically made from polypropylene (PP) due to their durability, chemical resistance, and transparency.

Figure 1. Sample cup

2.2 Design Procedure

Before conducting the design process, it is necessary to create a design flowchart. This helps illustrate the design process to be carried out, making it easier to understand. The flowchart is described in Figure 2. The flowchart outlines the design and simulation process for the Sample Cup injection molding using the Three Mold Plate method. The workflow begins with a literature study, followed by the identification of product specifications, including product dimensions and material selection. This initial step ensures that the design aligns with industry standards and functional requirements.

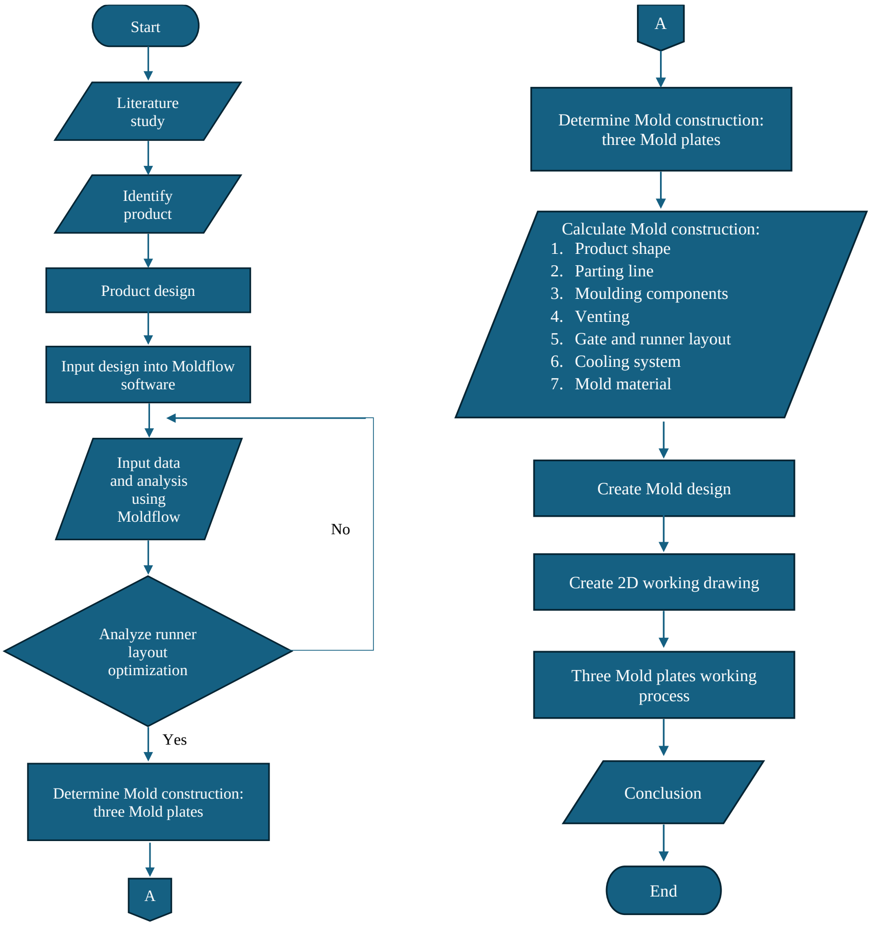


Figure 2. Design flowchart

Once the product specifications are determined, the design process is carried out in Autodesk Inventor Professional 2024. The designed model is then imported into Moldflow Plastic Insight for further analysis. At this stage, Moldflow Insight software is utilized to conduct a detailed simulation of key parameters, including [17], [18]:

1. Meshing element – ensuring a structured model for accurate simulation.
2. Gate location analysis – determining the best entry point for the molten plastic.
3. Injection location analysis – optimizing the injection position for uniform filling.
4. Molding window analysis – evaluating process feasibility under different conditions.
5. Fill analysis – examining how the plastic flows within the mold.
6. Runner system layout analysis – optimizing runner pathways to enhance efficiency.
7. Cooling system analysis – evaluating cooling time to prevent defects.

Following the simulation, the data analysis results provide insights into optimizing the runner system layout. This step ensures that material flow is balanced, minimizing defects such as air traps, weld lines, and warpage.

With the optimized runner system, the next phase involves mold construction design. The mold design process considers several critical factors:

1. Product shape and tolerances – ensuring dimensional accuracy.
2. Mold components and equipment – compatibility with injection molding machines.

3. Parting line design – determining mold separation points.
4. Venting – allowing trapped air to escape and preventing defects.
5. Number of cavities – optimizing production efficiency.
6. Gate and runner configuration – ensuring smooth material flow.
7. Injection system – defining the injection mechanism.
8. Cooling system – improving cycle time and product quality.
9. Mold material selection – choosing durable materials for long-term use.
10. Surface finishing – ensuring aesthetic and functional quality.

Once the mold design is finalized in Autodesk Inventor Professional 2024, a 2D technical drawing is created to guide manufacturing. The operational mechanism of the Three Mold Plate system is then analyzed to ensure proper functionality during the injection molding process.

The process concludes with a summary of findings, recommendations, and finalization of the injection molding design, ensuring that the Sample Cup meets industry standards for efficiency, durability, and quality.

3. RESULTS AND DISCUSSION

3.1 Material Selection

Polypropylene (Polyflam RPP1058UHF) is selected as the material for Sample Cups due to its excellent chemical resistance, mechanical properties, and suitability for injection molding. Polypropylene is highly resistant to acids, bases, alcohol, and most organic solvents, making it ideal for storing chemical samples without degradation or contamination [19]. Unlike some polymers, polypropylene absorbs minimal water, preventing chemical dilution or alteration of liquid samples. Polyflam RPP1058UHF is a high-purity polypropylene variant with low extractables and leachable, ensuring it does not release contaminants into stored substances. With a melt temperature of 168–200°C and crystallization point of 130–135°C, it maintains shape and integrity even under sterilization processes like autoclaving.

3.2 Gate Location

Figure 3 shows the gate location analysis for the sample cup injection molding process. The color scale represents the filling quality, where blue indicates the best conditions and red the worst conditions.

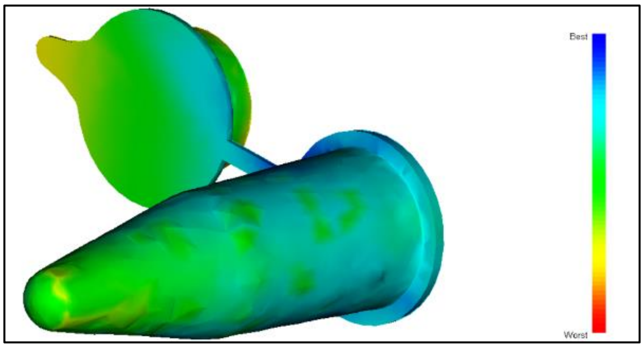


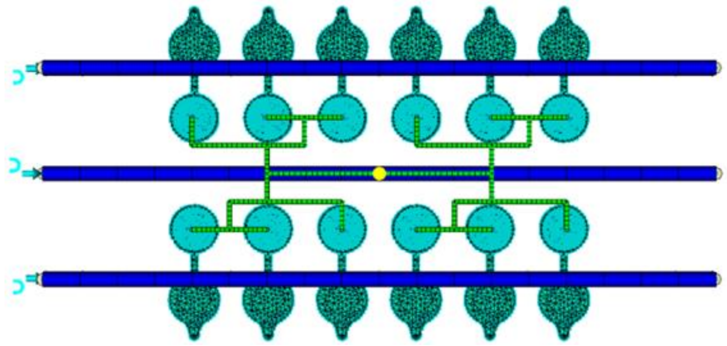
Figure 3. Gate location analysis

Most of the part appears blue, meaning uniform material flow, minimal defects, and good filling conditions. Despite the favorable flow pattern, the proposed gate location is not suitable due to mold design constraints. The gate location may interfere with the ejector pin placement, leading to difficulty in part removal. The gate mark may affect the sealing surface or functional areas of the sample cup. And moreover, the mold design might require complex modifications, increasing production costs and cycle time.

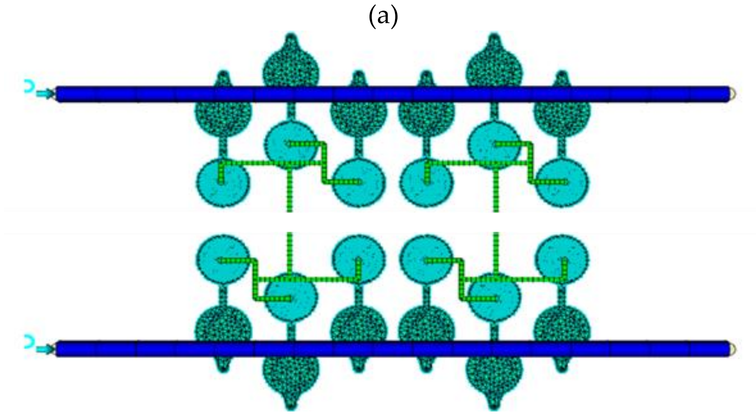
Considering the mold design constraints, the gate location is ultimately chosen at the tip of the product. Placing the gate at the tip ensures uniform material flow along the axis, reducing the risk of warping or deformation, and maintaining the cylindrical shape of the sample cup. The selected location allows for even material distribution, minimizing weld lines and air traps that could compromise the structural integrity. The gate position ensures better ejection, reduces mold complexity, and prevents aesthetic defects on critical surfaces.

3.3 Runner System

Two variations of the runner system layout will be evaluated through material flow simulation within the runner system, as described in Figure 3. The runner layout variations will be the focus of comparison to determine the most optimal runner layout for the injection molding process of the sample cup product. The primary objective of this comparison is to achieve optimal flow during the plastic filling process into the mold.



(a)



(b)

Figure 4. Two variations of runner system: a) Layout 1; b) Layout 2

The two runner system layouts, Layout 1 and Layout 2, are evaluated based on material flow efficiency and mold cavity filling uniformity.

- Layout 1 features a longer runner system with additional branching, which may result in increased pressure drop and longer filling time. However, it may provide better balance in flow distribution to all cavities.
- Layout 2 has a more compact runner design with reduced branching, minimizing pressure loss and ensuring faster injection. However, the shorter runner paths may lead to uneven filling between cavities.

The primary consideration for selecting the optimal layout is achieving the most balanced flow with minimal pressure drop, ensuring uniform filling and high-quality sample cup production.

3.4 Cooling System

The cooling system shown in Figure 5 consists of multiple parallel cooling channels, strategically placed to ensure efficient heat dissipation during the injection molding process.

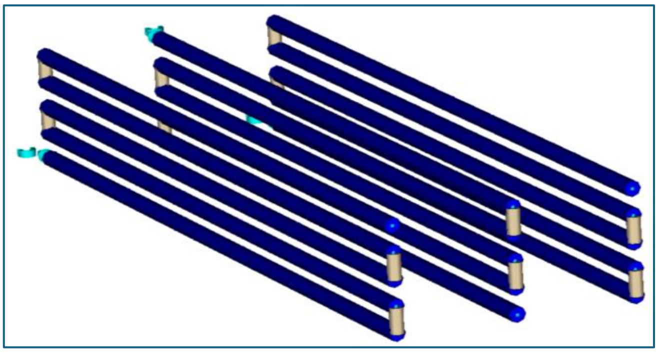


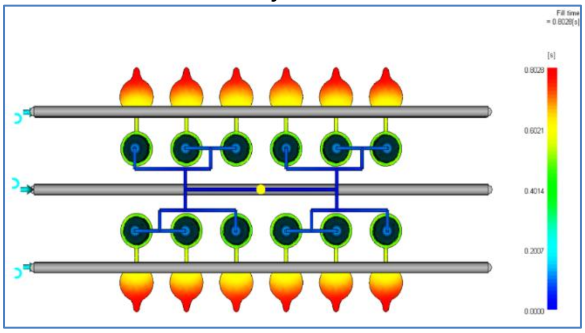
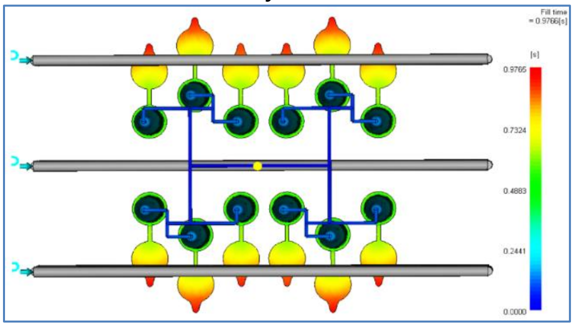
Figure 5. Cooling channels

Several key considerations for this cooling system design include:

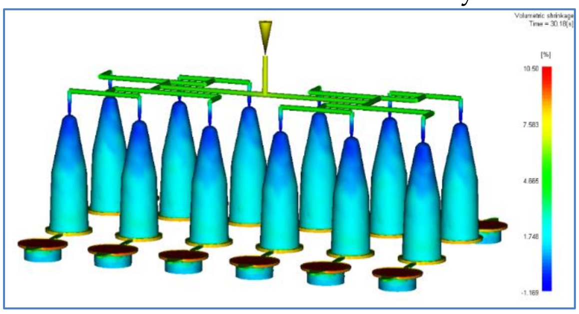
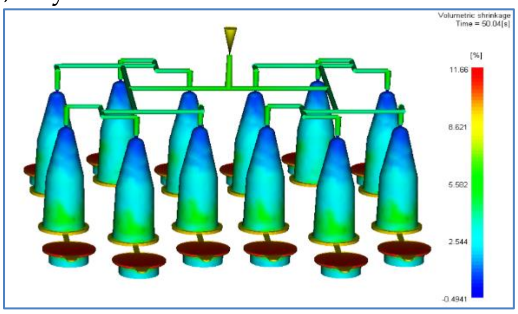
- The layout ensures consistent cooling across all mold cavities, reducing the risk of warpage and shrinkage in the final product.
- The parallel arrangement of cooling channels allows for faster heat removal, minimizing the mold cooling time and improving production efficiency.
- The cooling channels are evenly distributed to maintain a balanced temperature profile across the mold, preventing hotspots and ensuring dimensional accuracy of the sample cups.
- The design incorporates optimal flow rates and coolant temperature control to enhance thermal exchange and maintain a stable molding process.
- The placement of cooling channels avoids interference with critical mold components such as the runner system and ejection mechanism while maintaining the mold's structural integrity.

3.5. Analysis of Moldflow Simulation Results

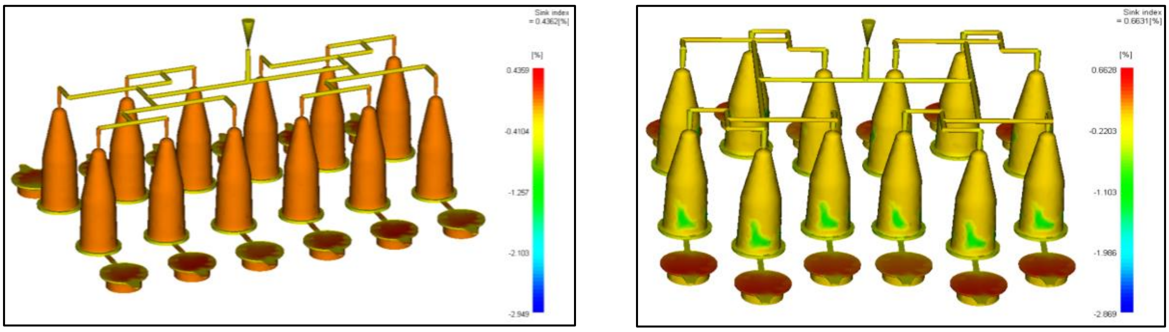
The following is a comparison of the Moldflow simulation results from two runner system layout variations.

Layout 1	**Layout 2**
	

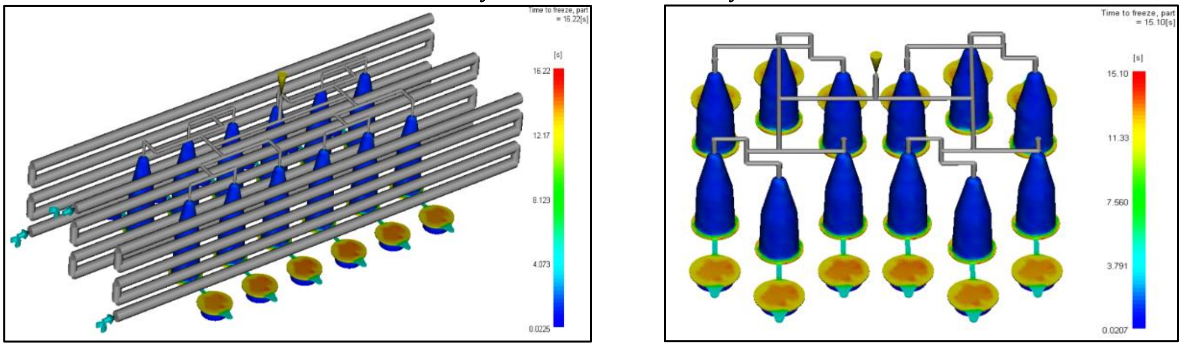
Fill time: Layout 1 - 0.81 s; Layout 2 – 0.97 s

Volumetric shrinkage: Layout 1 – 30.19%; Layout – 50.04%



Sink index: Layout 1 – 0.436%; Layout 2 – 0.663%



Time to freeze of parts: Layout 1 – 16.22s; Layout 2 – 15.1s

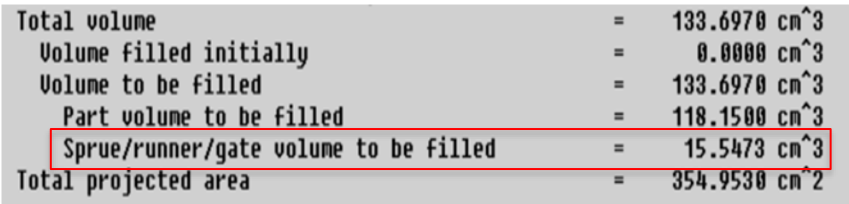
Figure 6. Moldflow analysis results

Based on the Moldflow simulation results shown in Figure 6, the following is a comparative analysis between Layout 1 and Layout 2:

- Layout 1 achieves a faster fill time, indicating a more efficient material flow. The reduced fill time helps minimize the risk of incomplete filling, flow hesitation, and potential defects such as weld lines. Layout 2, with a longer fill time, suggests a less efficient flow path, leading to increased cycle time and potential material degradation.
- Layout 1 exhibits significantly lower volumetric shrinkage compared to Layout 2. High shrinkage in Layout 2 can lead to greater warpage and dimensional instability, making it less suitable for high-precision components. The lower shrinkage in Layout 1 ensures better dimensional accuracy and structural integrity of the final product.
- The sink index measures the likelihood of surface defects, particularly sink marks. Layout 1 has a lower sink index, indicating a more uniform material distribution and better part aesthetics. In contrast, Layout 2 has a higher sink index, increasing the risk of visible surface defects, which may require additional post-processing or lead to rejection.
- Layout 1 has slightly longer freezing time, which allows for better material packing and reduces internal stress. While Layout 2 freezes faster, it may lead to higher residual stress and increased risk of warpage due to uneven cooling. The longer freezing time in Layout 1 contributes to improved part quality and mechanical stability.

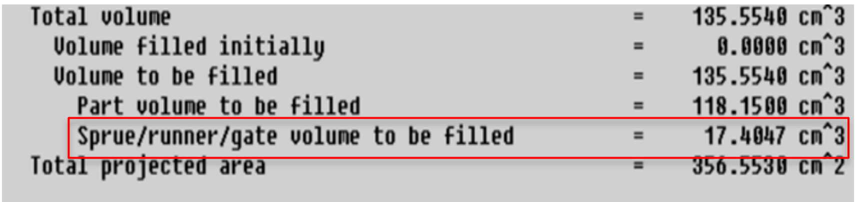
Overall, Layout 1 provides a more balanced and optimized injection molding process, making it the preferred runner layout for achieving high-quality and defect-free sample cups.

Additionally, based on the runner volume analysis in Figure 7, Layout 1 produces a runner volume of 15.55 cm^3 with a runner waste weight of 14 grams, whereas Layout 2 results in a runner volume of 17.4 cm^3 with a runner waste weight of 15.7 grams.



Total volume	=	133.6970 cm^3
Volume filled initially	=	0.0000 cm^3
Volume to be filled	=	133.6970 cm^3
Part volume to be filled	=	118.1500 cm^3
Sprue/runner/gate volume to be filled	=	15.5473 cm^3
Total projected area	=	354.9530 cm^2

(a)



Total volume	=	135.5540 cm^3
Volume filled initially	=	0.0000 cm^3
Volume to be filled	=	135.5540 cm^3
Part volume to be filled	=	118.1500 cm^3
Sprue/runner/gate volume to be filled	=	17.4047 cm^3
Total projected area	=	356.5530 cm^2

(b)

Figure 7. Runner volume

3.6 Mold Design

Based on the Moldflow simulation results, Runner Layout 1 was chosen as it demonstrated more optimal performance compared to Layout 2, although the difference was not highly significant. Additionally, considerations such as product placement for ease of mold fabrication were also considered. The mold design process begins with determining the basic mold dimensions based on the sample cup size and the manufacturing specifications of Futaba and Misumi. Moldflow Insight analysis was utilized to design the runner system, and the results of this analysis were then used to develop the Cavity and Core Plate using the Futaba DA-S-IH 500 mm x 500 mm mold, which employs a three-plate mold technique [20]. The mold material selected is S55C, due to its high strength and resistance to high temperatures. The image below illustrates the three-plate mold assembly design along with 2D drawings of the mold base components. Figure 8 shows the 3D model result of the mold assembly for the sample cup product, while, Figure 9 present the components of the mold.

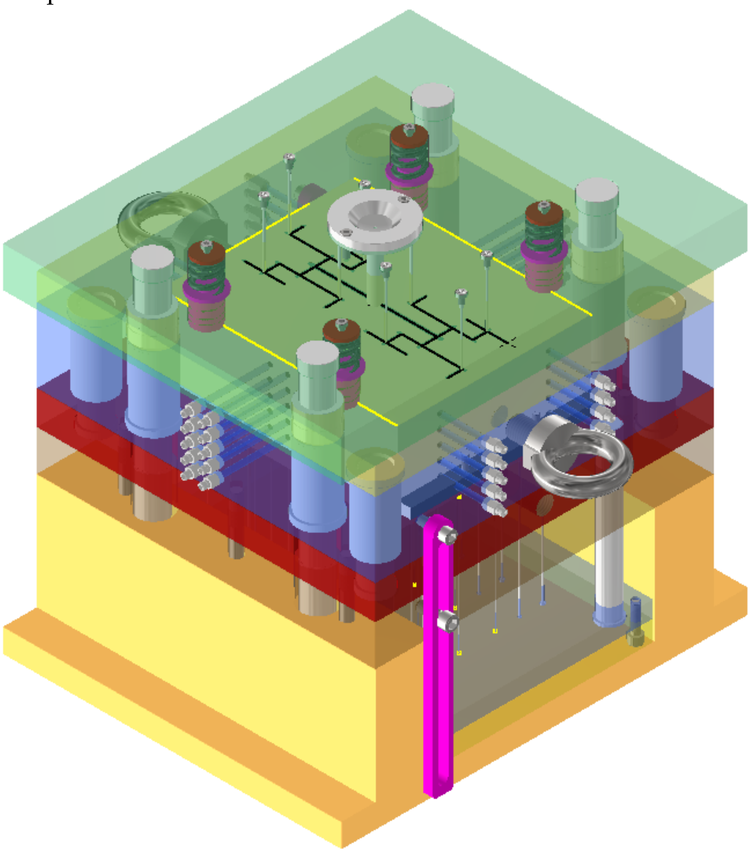


Figure 8. 3D model of mold assembly

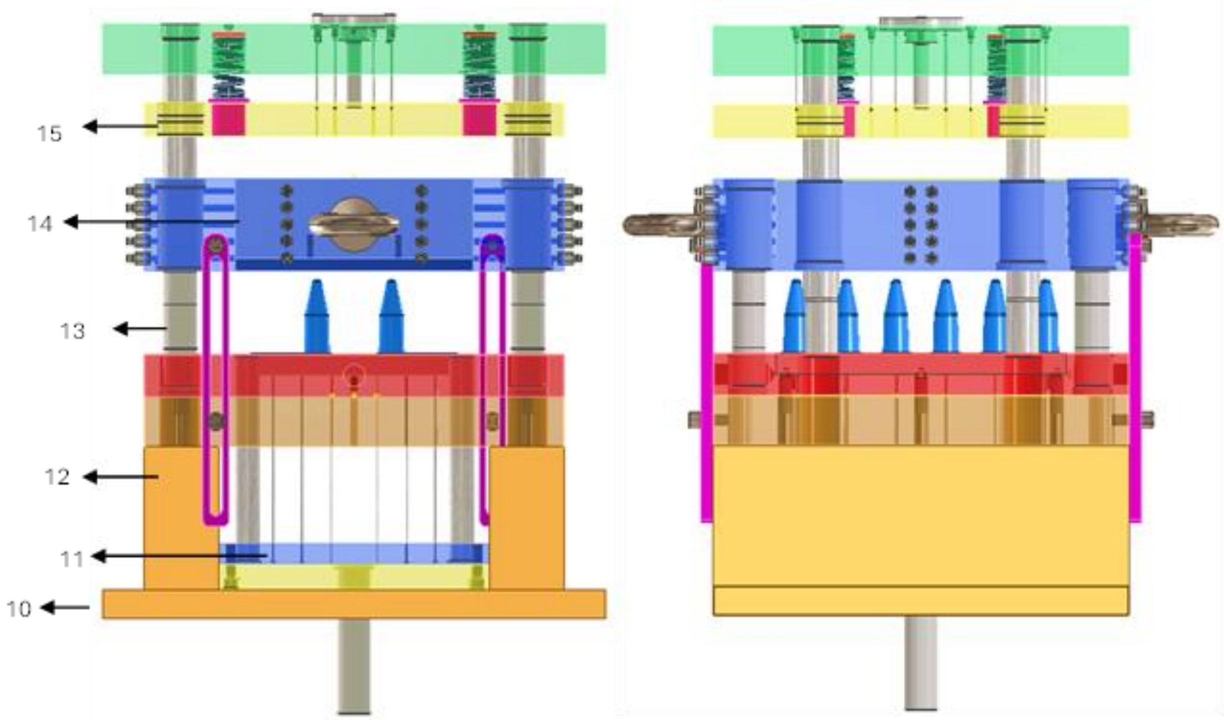


Figure 9. Mold components

Table 1 shows the recommended material selection for each mold component based on JIS standards and its function in the mold assembly [21]

Table 1. Part list and material selection

No	Part Name	Recommended Material (JIS)	Reasoning
1	TOP CLAMPING PLATE	S50C / S55C	High strength and resistant to pressure
2	RUNNER STRIPPER PLATE	S50C / S55C	Durable and withstands pressure during injection process
3	GUIDE BUSH	SKD61	Wear-resistant and has good heat resistance
4	SUPPORT PIN	SKD11	High wear resistance and excellent strength
5	CORE PLATE	S50C / S55C	Withstands pressure and is easy to machine
6	SUPPORT PLATE	S50C	Provides structural stability to the mold
7	RETURN PIN	SKD11	High wear resistance and durability
8	EJECTOR RETAINER PLATE	SKD61	Resistant to friction and high temperatures
9	Cylinder Head Cap Screw	SCM435	High tensile strength alloy steel
10	BOTTOM CLAMPING PLATE	S50C / S55C	Strong and stable for holding the mold load
11	CORE PLATE	S50C / S55C	Main mold structure with good strength
12	SPACER BLOCK	S50C	Used for mold height adjustment
13	GUIDE PIN	SKD11	High resistance to friction and wear
14	CAVITY PLATE	SKD61	High heat resistance and wear-resistant
15	GUIDE BUSH	SKD61	Highly durable against friction and heat

4. CONCLUSION

Based on the Moldflow simulation, Runner Layout 1 was chosen for its better performance despite minor differences from Layout 2. It had shorter fill time (0.81s vs. 0.97s), lower shrinkage (30.19% vs. 50.04%), reduced sink index (0.437% vs. 0.663%), and lower runner waste (14g vs. 15.7g), making it more efficient.

The mold design process began with determining the basic mold dimensions based on the sample cup product size and manufacturing standards from Futaba and Misumi. The Moldflow Insight analysis was crucial in designing the runner system, and the results were used to develop the Cavity and Core Plate (Futaba DA-S-IH 500 mm × 500 mm), which utilizes a three-plate mold technique. The selected mold materials were chosen for their high strength and heat resistance, ensuring durability and prolonged mold life.

This research demonstrates that careful runner system optimization and mold material selection play a crucial role in improving the injection molding process efficiency, reducing defects, and minimizing material waste.

REFERENCES

[1] D. H. Chun, "Numerical analysis of injection molding for the Syringe Barrel with optimum design and processing condition," *Fibers and Polymers*, vol. 18, no. 9, pp. 1790–1795, 2017, doi: 10.1007/s12221-017-7065-3.

[2] A. K. Mufid *et al.*, "Perancangan Injection Molding Dengan Sistem Three Plate Mold Pada Produk Glove Box," *JMPM: Jurnal Material dan Proses Manufaktur*, vol. 1, no. 2, pp. 72–81, 2017.

[3] M. Chien, Y. Lin, C. Huang, and S. Hwang, "Impact of Runner Size, Gate Size, Polymer Viscosity, and Molding Process on Filling Imbalance in Geometrically Balanced Multi-Cavity Injection Molding," *Polymers (Basel)*, vol. 16, no. 20, Oct. 2024, doi: 10.3390/polym16202874.

[4] S. Kitayama, H. Miyakawa, M. Takano, and S. Aiba, "Multi-objective optimization of injection molding process parameters for short cycle time and warpage reduction using conformal cooling channel," *International Journal of Advanced Manufacturing Technology*, vol. 88, no. 5–8, pp. 1735–1744, Feb. 2017, doi: 10.1007/s00170-016-8904-x.

[5] L. Khoirul Miftakhul Ni, C. Budiyantoro, M. Budi Nur Rahman, T. Mesin, and U. Muhammadiyah Yogyakarta Jl Lingkar Selatan Tamantirto, "DESAIN DAN OPTIMASI INJECTION MOLD SISTEM SLIDER PADA PRODUK PREFORM STICK T15," *Jurnal Rekayasa Mesin*, vol. 8, no. 3, pp. 155–165, 2017.

[6] K. Monkova, P. Monka, A. Cizikova, and S. Hric, "Mould running system design to achieve the minimum waste," *Adv Mat Res*, vol. 933, pp. 963–968, 2014, doi: 10.4028/www.scientific.net/AMR.933.963.

[7] D. O. Kazmer, *Injection Mold Design Engineering*. 2007. doi: 10.3139/9783446434196.fm.

[8] A. K. Mufid, C. Budiyantoro, M. Budi, and N. Rahman, "PERANCANGAN INJECTION MOLDING DENGAN SISTEM THREE PLATE MOLD PADA PRODUK GLOVE BOX," 2017. [Online]. Available: http://journal.umy.ac.id/index.php/jmpm

[9] M. Moayyedian, K. Abhary, and R. Marian, "The analysis of short shot possibility in injection molding process," *International Journal of Advanced Manufacturing Technology*, vol. 91, no. 9–12, pp. 3977–3989, 2017, doi: 10.1007/s00170-017-0055-1.

[10] C. Budiyantoro, *Thermoplastik Dalam Industri*. Yogyakarta: Teknika Media, 2009.

[11] M. Choirul Anwar, C. Budiyantoro, and T. Thoharudin, "Optimalisasi Parameter Proses Injeksi Menggunakan Simulasi Moldflow untuk Meminimalkan Cycle Time dan Eliminasi Short Shot pada Produk Tempat," *JMPM (Jurnal Material dan Proses Manufaktur)*, vol. 2, no. 1, 2018, doi: 10.18196/jmpm.2121.

[12] R. Baesso, M. Salvador, and G. Lucchetta, "Filling balance optimization for plastics injection molding," in *CISM International Centre for Mechanical Sciences, Courses and Lectures*, vol. 486, Springer International Publishing, 2005, pp. 617–624. doi: 10.1007/3-211-38053-1_60.

[13] M. Szczęsna, M. Szindler, J. Weszka, M. Szindler, and A. Żebracka, "Determining the melt flow index of polypropylene: Vistalon 404," *Journal of Achievements in Materials and Manufacturing Engineering*, vol. 61, no. 2, pp. 308–314, 2013.

[14] I. Toledo Pimentel, N. Isabel, A. Acevedo, M. Cristina, G. Rocha, and -Mrocha@iprj Uerj Br, "Rheological and Mechanical Properties of Recycled Polypropylene / Virgin Polypropylene Blends," 2018.

[15] Chandra Asri, "Trilene Homopolymer Polypropylene HI10HO," Jakarta, 2021.

[16] J. S. S. Han, "A study on reduction of clamping force for plastic back cover of large TV," vol. 13, no. 1, pp. 36–41, 2019.

[17] H. B. Wang, Z. R. Li, C. H. Sun, and Y. P. Zhang, "Runner balance design of soap box compounding cavity based on moldflow," *Adv Mat Res*, vol. 941–944, pp. 1678–1681, 2014, doi: 10.4028/www.scientific.net/AMR.941-944.1678.

[18] H. Wang and C. Sun, “Optimization Analysis of Correction Tape Box Compounding Cavity Runner Balance Based on Moldflow,” *World Journal of Engineering and Technology*, vol. 08, no. 01, pp. 42–49, 2020, doi: 10.4236/wjet.2020.81005.
[19] A. . Schulman, “POLYFLAM TM RPP 1058 General Purpose Homopolymer Polypropylene (30 % Fiberglass) Contact Songhan Plastic Technology Co ., Ltd .,” 2022.
[20] Futaba, “Futaba Standard Plastic Mold Components, Blue Book,” Chiba, 2020.
[21] JFE Steel Corporation, “SPECIAL STEEL SHEET,” Chiyodaku, 2022.