Injection Moulding Design, Simulation and Analysis Using SolidWorks and MoldFlow

Azeim Muhamad[1], Mohd Yussni Hashim[*1]

[1] *Faculty Of Mechanical Engineering*
Universiti Tun Hussein Onn Malaysia, 86400, Parit Raja, Batu Pahat, MALAYSIA

*Corresponding Author: yusni@uthm.edu.my
DOI: https://doi.org/10.30880/rpmme.2025.06.01.012

Article Info	Abstract
	This study analyzes and optimizes two injection molding runner designs using SolidWorks and Moldflow simulation software. It compares key metrics like fill time, pressure distribution, material flow, and defects such as sink marks and warping. The results show that a simpler, shorter runner design, Design 1, outperforms a more complex, longer design, Design 2. Design 1 achieved a shorter fill time of 1.362 seconds compared to 1.407 seconds for Design 2, and a lower pressure drop of 113.6 MPa compared to 147.0 MPa for Design 2. Additionally, it showed better material flow and fewer defects. The study highlights the importance of using design and simulation tools to optimize injection molding, offering valuable insights for industries like automotive, electronics, and medical devices.

1. Introduction

Injection moulding is a widely used manufacturing process for creating parts from materials like thermoplastics and thermosets. It is commonly used in industries to produce products of varying sizes and complexities, including thin-walled structures like plastic housings. These housings are essential in items like power tools, household appliances, and electronics. Injection moulding is also used for everyday products, such as toothbrushes, combs, syringes, and valves.

The process involves three main components: a mould, raw material, and an injection moulding machine. Careful control of parameters like filling time, mould temperature, gate type, melt temperature, packing pressure, and packing time is crucial to avoid defects. Software like Moldflow can simulate the process and provide insights into potential issues like warpage, shrinkage, and air traps.

The challenges in the design of the runner system, which is crucial for the flow of molten material into the mold. A poorly designed runner system can lead to defects, inefficient production, and increased costs. The study aims to use simulation software like Moldflow and SolidWorks to analyze and improve runner designs, focusing on enhancing material flow, reducing defects, and improving efficiency. The objectives of the study is to analyze the impact of runner design on the quality of the final product using Moldflow software and to develop an optimal mold design using SolidWorks software, by focusing on design of runner.

The design of the runner system in injection molding is crucial for product quality, affecting factors like fill time, material usage, defects, and production efficiency. Studies show that optimizing runner size, layout, and injection points can improve filling and reduce issues like air traps and weld lines. This research uses Moldflow and SolidWorks to analyze and enhance runner design, aiming to reduce defects like warping and sink marks

while improving material flow and efficiency. By combining both tools, the study demonstrates how they can optimize injection molding processes, reduce waste, and increase productivity [1], [2].

This research focuses on integrating sustainable practices into injection molding systems by optimizing the runner design using SolidWorks and Moldflow. The study aims to reduce material waste, energy consumption, and production inefficiencies. By refining the runner system, unnecessary extensions are eliminated, leading to a more efficient flow of materials, faster cavity filling, and lower energy use per cycle. Moldflow simulations help prevent defects like air traps and weld lines, improving product quality and reducing material waste. Additionally, using simulation tools allows for virtual testing before physical molds are made, saving time and resources. Optimizing the runner system also reduces mold stress, extending its lifespan and lowering environmental impact. Ultimately, the study promotes sustainable manufacturing practices by cutting waste, improving efficiency, and supporting a circular economy [3], [4], [5].

2. Methodology

Methodology refers to the systematic and theoretical analysis method applied to a specific field of study. In this chapter, the entire process and procedures of the project will be discussed. Generally, the method used in this project involved modelling of the product using solidworks, simulation and analysis using Moldflow software and using statistical and data analysis tool will be used to optimize the data.The process for improving runner design in injection molding follows a clear, step-by-step approach to ensure the design meets the project's goals. First, set clear objectives, such as ensuring even material flow, reducing defects, and adhering to size and material constraints. This helps stay focused throughout the process.

Next, create the initial design of the mold and runner system in SolidWorks, deciding on the runner's size, length, and gate positions. These details are critical for proper material flow. Once the design is complete, it's transferred to Moldflow for simulation. In Moldflow, set up a simulation by entering the material type and process conditions like pressure and temperature. The simulation helps identify potential problems, such as uneven flow or air traps. If any issues are found, adjust the design and test again. This process is repeated until an optimal design is achieved.

Once the design is finalized, compare it to the original to assess improvements. Finally, prepare a report summarizing the entire process, the changes made, and the results. This report serves as a useful reference for future projects, ensuring the final design is effective, efficient, and meets the project's goals
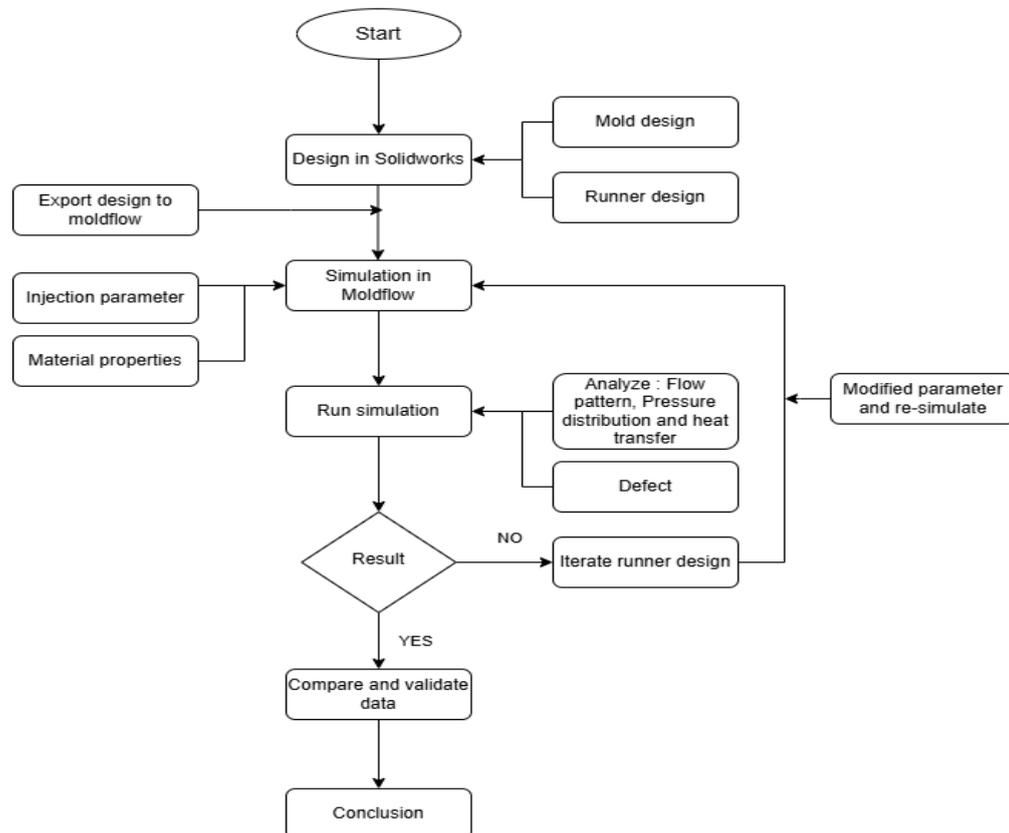

Fig1 *Methodology Flow Chart*

2.1 Mold Design in SolidWorks

Mold design is crucial in injection molding as it directly affects product quality, including part geometry, dimensional accuracy, surface finish, and mechanical properties. It ensures proper molten plastic flow, minimizes defects like warping and voids, and optimizes cooling channels, ejector systems, and gate placement. Additionally, selecting the right materials for durability is essential. Overall, effective mold design requires expertise to enhance efficiency, cost-effectiveness, and success in injection molding.

2.1.1 Cavity and Core Design

The cavity in mold design is responsible for shaping the external surface of the molded part, where molten material is injected and cooled to form the desired product. In SolidWorks, the cavity design begins by defining the shape of the part and extracting the negative geometry from the 3D model. To account for material shrinkage and ensure proper ejection of the molded part, the cavity is typically created by offsetting the part geometry. This process ensures that the final product fits the desired specifications.

The mold's exterior dimensions are carefully planned for both design Layout 1 and Layout 2 in Figure 2, is to maintain structural integrity and house the cavities effectively. These dimensions are 480.92 mm in length, 291.15 mm in width, and 30 mm in height. These precise measurements are essential to providing a sturdy framework that supports consistent and uniform heat transfer throughout the mold, which is crucial for efficient molding.
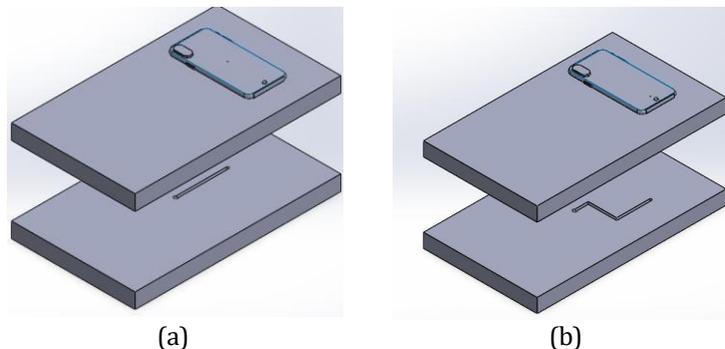

(a)   (b)

Fig 2: *(a) Cavity for Layout 1, (b) Cavity for Layout 2*

The core design is matches the cavity dimensions with a length of 480.92 mm, width of 291.15 mm, and height of 30 mm. This consistency between the core and cavity ensures a seamless fit, enhancing the overall efficiency of the molding process. Inside the core, as marked by the red box in Figure 3, are the product dimensions. The product's size at the core is 160.31 mm in length, 81.15 mm in width, and 7.7 mm in height, which corresponds to the external dimensions of the final product. The core design is essential because it enables the material to flow evenly around the core, filling the cavity consistently and maintaining the shape of the product. With these accurate core dimensions, the material fills the mold evenly, which improves both the quality and speed of the production process, ensuring a more effective and reliable outcome.
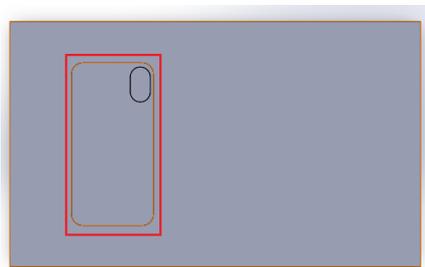

Fig 3: *The core mold*

2.1.2 Gating Design

The gating system in injection molding is crucial for ensuring high-quality parts and efficient production. The pin-point gate, used in this process, controls the flow of molten material into the mold cavity. It is ideal for small, detailed parts, preventing overpacking and ensuring proper filling. This gate type is also suitable for three-plate molds, used for part separation and ejection. Its small, sharp opening allows for controlled material flow, reducing waste.

In Design 1, the gating position is centrally placed in the cavity, promoting an even material flow and uniform fill. The shorter runner reduces the material's travel distance, making the fill faster and more efficient. However, in Design 2, the gating is placed on the side of the cavity, allowing better control of material flow, especially for complex or asymmetrical cavity designs. The longer runner introduces more resistance, potentially slowing the fill, but side gating ensures precise material distribution, which is vital for intricate designs. Figure 4 below shown the gating position in cavity.
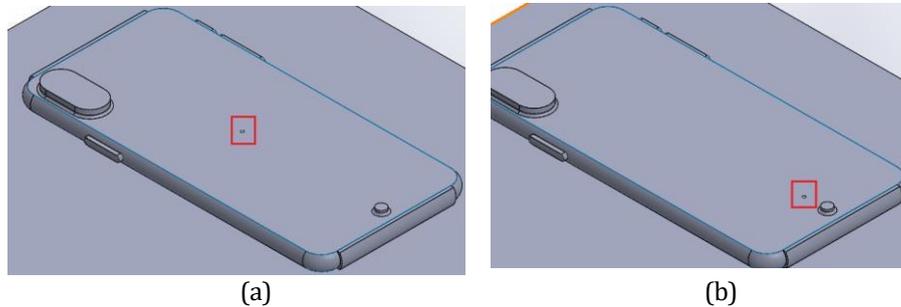

(a)   (b)

Fig 4: *The gating position in cavity*

2.1.3 Runner Design

In injection molding, the runner system is essential for directing molten material from the nozzle to the mold cavity through the gate. The runner design affects material flow, fill speed, and product quality. Two runner layouts are compared to study the impact of runner length and gate position. Both layouts use a pin-point gate to control material flow.

Layout 1, shown in Table 1, is 135.2 mm long and features a straight section with a turn at the bottom, creating a shorter, more efficient path for the material. In contrast, Layout 2, shown in Table 1, is 197.7 mm long and more complex, with several turns and a longer runner. This layout is suited for molds with complex shapes but is less efficient due to increased cooling time, material waste, and pressure drops. Both layouts have the same 5mm diameter and similar basic structure, but Layout 1 is more efficient due to its shorter, simpler design, reducing material waste and speeding up the process. Layout 2, though offering better flow control, leads to more waste and longer cycle times.

Table 1 *Parameter of the Runner*

Runner Layout	Length(mm) (t)	Diameter(mm) (t)
Layout 1	135.2	5
Layout 2	197.7	5

2.2 Simulation Using Moldflow

Moldflow simulation software is essential for modeling and analyzing the injection molding process. It helps predict potential manufacturing issues and optimizes mold design and material selection before production begins. By simulating material behavior within the mold cavity, Moldflow identifies defects such as warping, air entrapment, and flow lines, ensuring better product performance.

The first step in using Moldflow Adviser software is exporting the phone case model from SolidWorks. This allows simulation of the runner system and material flow to ensure uniform mold filling and efficient operation. The selected process sequence in Moldflow Adviser includes fill + pack, cool, sink, and warp. The results provide insights into fill confidence, quality prediction, fill time, injection pressure, flow front temperature, sink depth, and deflection, helping to optimize the design process.

The selection of material and its properties are the crucial to simulating the injection molding process accurately. In this study, the material selected is polycarbonate (PC), known for its combination of strength and durability. The viscosity, density, thermal conductivity, specific heat and shrinkage are automatically set as the material selected in the Moldflow Adviser software. The figure below shown the material selection in Moldflow Adviser software.

2.2.1 Parameter in Moldflow

Injection pressure is a key parameter that controls material flow into the mold. In this simulation, the injection pressure is set to 300 MPa, based on the machine specifications and PC material flow characteristics, ensuring proper mold filling and preventing defects. Temperature also plays a vital role; the melt temperature is set to 260°C for smooth material flow, while the mold temperature is set at 95°C to maintain the right viscosity and avoid premature cooling, which can cause defects.

The injection time is set to automatic in Moldflow Adviser, while the injection hold time is set to 4 seconds. The injection time ensures efficient mold filling without excessive shear stress, and the hold time compensates for material shrinkage during cooling. The Table 2 below shows the processing parameter in Moldflow Adviser software.

Table 2 *Processing parameter in Moldflow*

Processing parameter	Value
Injection pressure	300 MPa
Melt temperature	260°C
Mold temperature	95°C
Injection hold time	4s

After conducting the simulation, the data was obtained. Based on this data, comparisons were made to assess two type of runner layout using the same parameter in the plastic injection molding process. Four comparisons were conducted. First, the results of the fill time were compared for both runner layout. Secondly, for pressure distribution for both runner layout. Next, for it material flow and defect for both the runner. Verifying the data is crucial to determine if the project objectives have been met.

3. Result and Discussion

This section presents the results of the simulation analysis conducted using SolidWorks for designing the phone case mold and Moldflow Adviser for simulating the runner system. These tools were used together to study the impact of runner design changes on product quality and to identify the best mold design.

The study aimed to understand how runner design affects molded product quality and to improve the mold design by focusing on the runner system. The results shared in this chapter are directly related to these goals, highlighting key findings and explaining the role of runner design in the injection molding process.

3.1 Analysis of Runner Design

In the study, two different runner designs were analyzed using Moldflow Adviser to assess their impact on the product quality and mold performance for a phone case. The first model featured a simple linear runner design, while the second model used a more complex runner design, with differences in runner length and gate position.

Both models shared the same material settings: a melt temperature of 260°C, a mold temperature of 95°C, and a maximum injection pressure of 300 MPa. The injection time was set to automatic, and the packing profile was consistent for both designs, ensuring that any differences in the results were due to the runner design variations.

3.1.1 Analysis for Runner Design 1

For the first design is a simple runner design. This runner layout underwent analysis through the Moldflow software. Table 3 shows the fill properties of the simulated mold design with runner layout 1. It takes 1.36 s to fill. For the shot volume is 33.6129cm3, while the cavity volume is 29.4639 cm3. Runner design 1 has a cycle time of 34.00 s. The warpage for runner design 1 is also shown in Table 3, the result for the nominal maximum deflection is 0.99 mm.

Table 3 *The properties from simulation result Design1*

Parameter	Value
Actual filling time (s)	1.36
Actual injection pressure (MPa)	113.627
Clamp force area (cm^2)	138.989

Max. clamp force during filling (tonne)	58.169
Velocity/pressure switch-over at % volume (%)	97.23
Velocity/pressure switch-over at time (s)	1.27
Total part weight at the end of filling (g)	34.063
Shot volume (cm^3)	33.612
Cavity volume (cm^3)	29.4639
Runner system volume (cm^3)	4.1490
Cycle time	34.00
Nominal max. deflection (mm)	0.99

The Moldflow simulation for Runner Design 1 shows good results in terms of efficiency and part quality. The total fill time is 1.362 seconds, which shows that the runner layout is effective in directing the molten material into the mold cavity. The fill time simulation also shows a uniform color gradient, meaning that the material flows evenly and fills all parts of the cavity without delays.
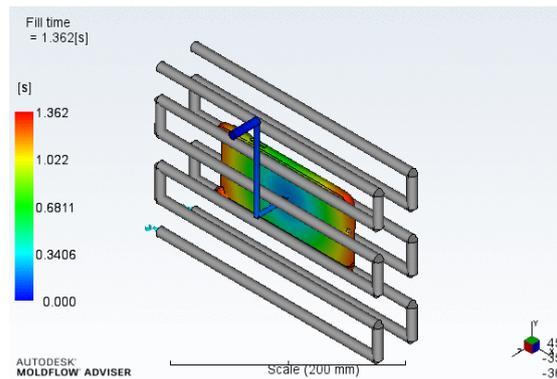

Fig 5 *Fill time for runner design 1*

The pressure drop throughout the runner and mold cavity for Runner Design 1 is measured at 113.6 MPa. The simulation reveals that the pressure distribution is relatively uniform, with no significant areas of high or low pressure. This consistent pressure aids in maintaining the quality of the molded part by avoiding uneven material packing, which can lead to defects such as warping or sink marks.
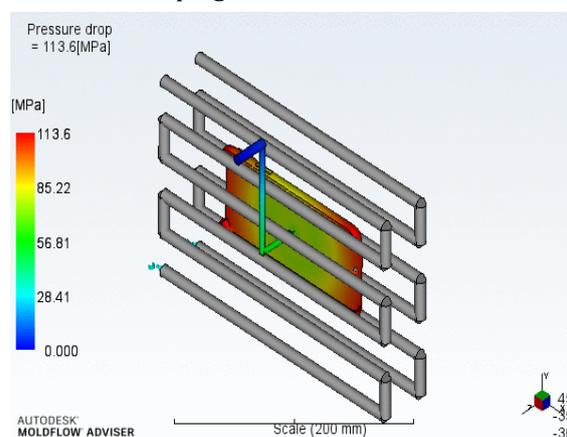

Fig 6 *Pressure distribution of runner design 1*

The plastic flow simulation indicates that the molten material moves evenly through the runner system and fills the mold cavity without any sudden changes or turbulence. This smooth flow helps to avoid common problems like air traps or weld lines. The confidence of fill results shows 100% fill confidence across the entire part, meaning the material reaches all areas of the cavity without any difficulty. Additionally, the fill time simulation shows that the material fills the cavity evenly and without delays, which helps maintain the uniformity and quality of the molded part. Overall, the material flow behavior in Runner Design 1 is effective and consistent, ensuring good part quality with minimal defects related to flow issues.
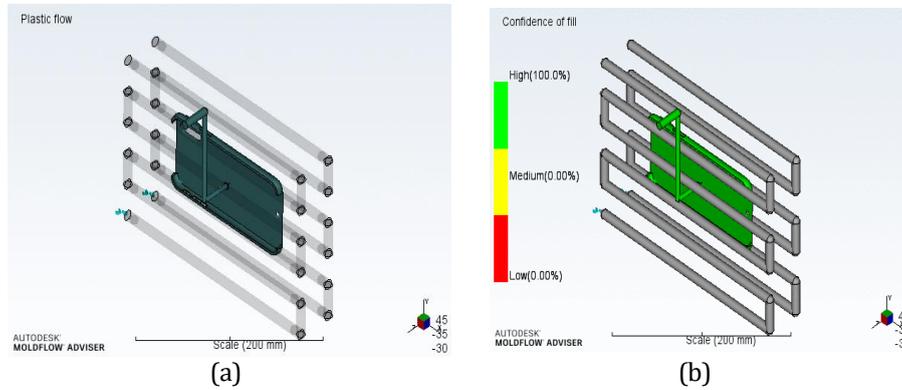

(a)   (b)

Fig 7 *(a) plastic flow and (b) confident of fill for runner design 1*

The analysis identified potential areas for air traps and weld lines. There is an couple minor air traps are present near sharp corners or areas with abrupt changes in flow direction. Weld lines were identified at specific locations where flow fronts meet, which is within acceptable limits. These defects are not expected to significantly affect the overall quality of the part but should be monitored for further improvement in future iterations of the design.
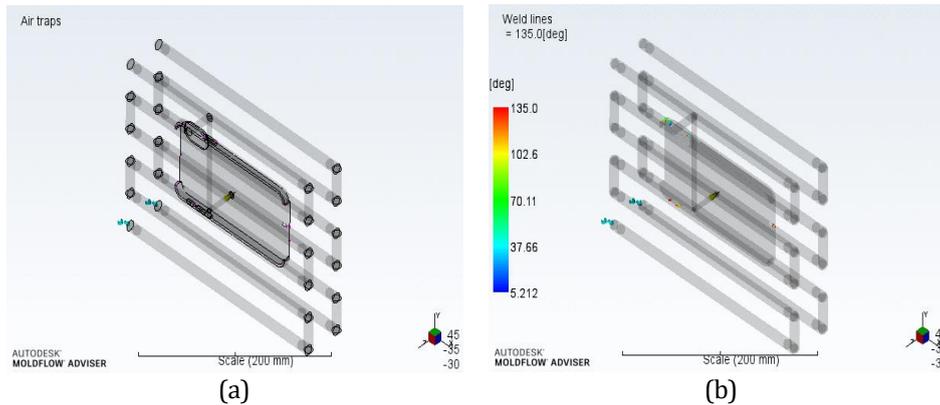

(a)   (b)

Fig 8 *(a)air trap and (b)weld line in Runner Design 1*

The runner design plays a significant role in ensuring the uniformity and dimensional accuracy of the molded part. The short and straightforward runner path in Runner Design 1 contributes to uniform pressure distribution and smooth material flow, reducing the likelihood of shrinkage and warping. The volumetric shrinkage at ejection is minimal, measured at 4.454%, indicating that the part maintains its shape and size within acceptable tolerances. Additionally, the sink marks are shallow, with an average depth of 0.0224 mm, demonstrating good material packing and cooling.
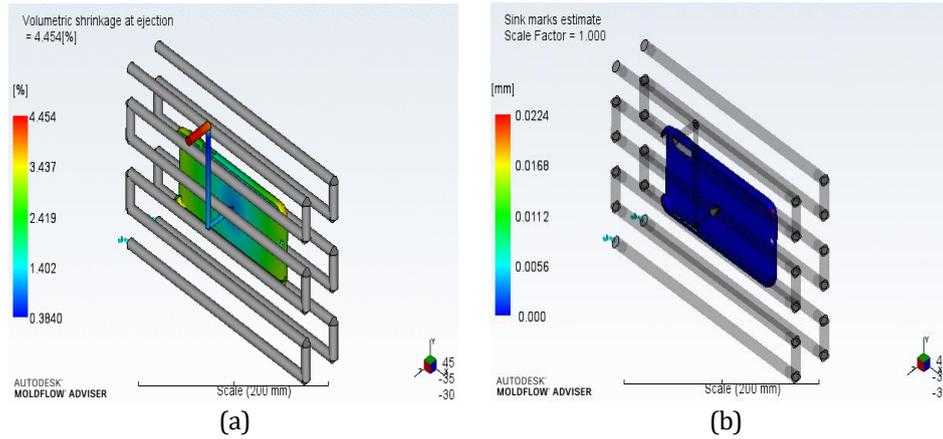
Fig 9 (a*)volumetric shrinkage and (b)sink mark for Runner Design 1*

3.1.2 Analysis Runner Design 2

For the second design is a more complex runner design. This runner layout underwent analysis through the Moldflow software. Table 4 shows the fill properties of the simulated mold design with runner layout 2. It takes 1.41s to fill. For the shot volume is 34.7930 cm3, while the cavity volume is 29.4649 cm3. Runner design 2 has a cycle time of 34.00 s. The warpage for runner design 2 is also shown in Table 4, the result for the nominal maximum deflection is 0.99 mm.

Table 4 *The properties from simulation result Design2*

Parameter	Value
Actual filling time (s)	1.41
Actual injection pressure (MPa)	146.968
Clamp force area (cm^2)	142.001
Max. clamp force during filling (tonne)	59.369
Velocity/pressure switch-over at % volume (%)	97.05
Velocity/pressure switch-over at time (s)	1.33
Total part weight at the end of filling (g)	34.212
Shot volume (cm^3)	34.793
Cavity volume (cm^3)	29.464
Runner system volume (cm^3)	5.328
Cycle time	34.00
Nominal max. deflection (mm)	0.99

The Moldflow simulation for Runner Design 2 shows a fill time of 1.41 seconds. The slightly longer fill time compared to Runner Design 1 can be attributed to the increased complexity and length of the runner system. The material takes more time to flow through the additional turns, which increases the overall fill time. Despite this, the fill time remains within an acceptable range for efficient part production.
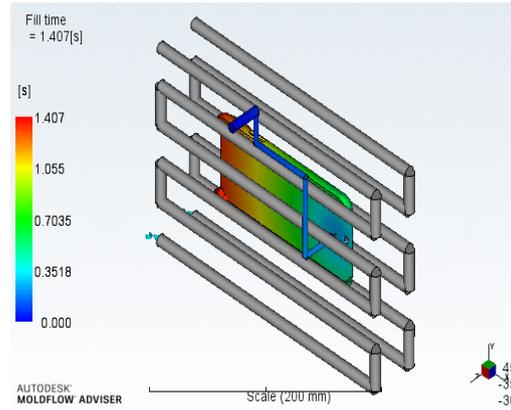
Fig 10 *Fill time of runner design 2*

The pressure drop observed in Runner Design 2 is 147.0 MPa, which is higher compared to Runner Design 1. The longer runner path and additional bends cause greater resistance to material flow, leading to this higher pressure drop. The pressure distribution is less uniform than in Runner Design 1, which can increase the likelihood of defects related to uneven material packing.
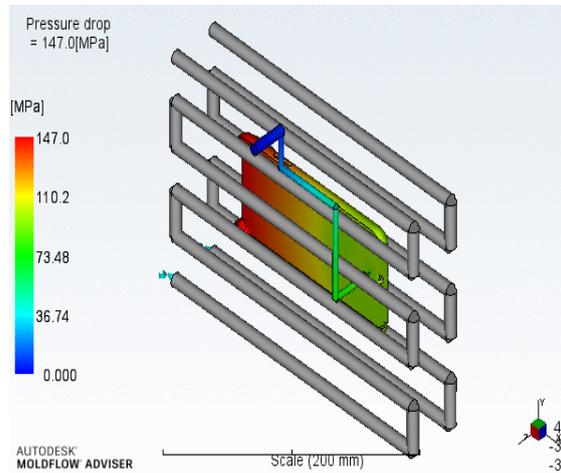

Fig 11 *Pressure distribution of runner design 2*

The plastic flow simulation indicates smooth material movement throughout the mold cavity; however, minor disturbances are observed near the runner bends due to the longer flow path. These disturbances could lead to slight variations in flow speed and increase the chances of turbulence. Despite these challenges, the material manages to fill the cavity adequately, as confirmed by the confidence of fill simulation, which shows 100% confidence for complete filling.
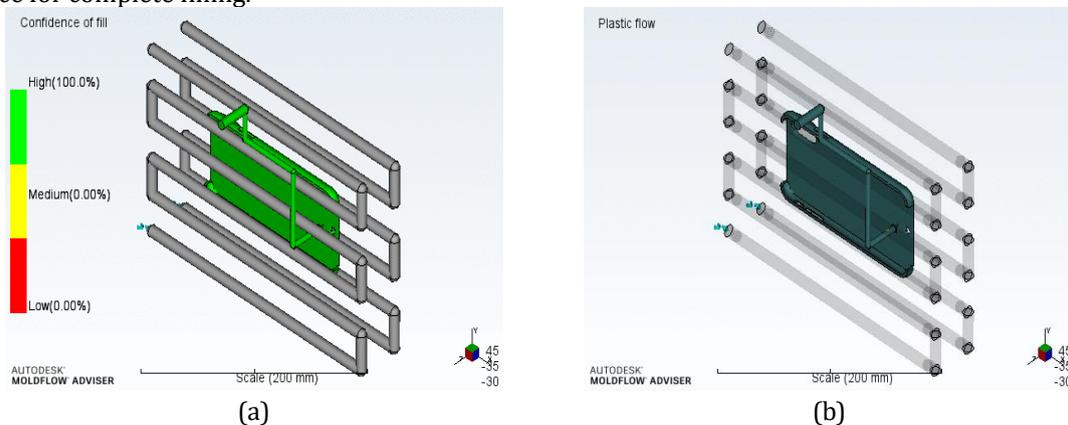

(a)   (b)

Fig 12: *(a)plastic flow and (b) confident of fill for runner design 2*

Several defects are observed in Runner Design 2. Minor air traps are detected near sharp corners and areas with abrupt flow changes. which is acceptable but may still pose minor quality concerns. No short shots are detected, indicating that the material reaches all areas of the cavity without major issues. Additionally, sink

marks are observed, with a maximum depth of 0.0258 mm, which is slightly lower than in Runner Design 1 and remains within acceptable limits
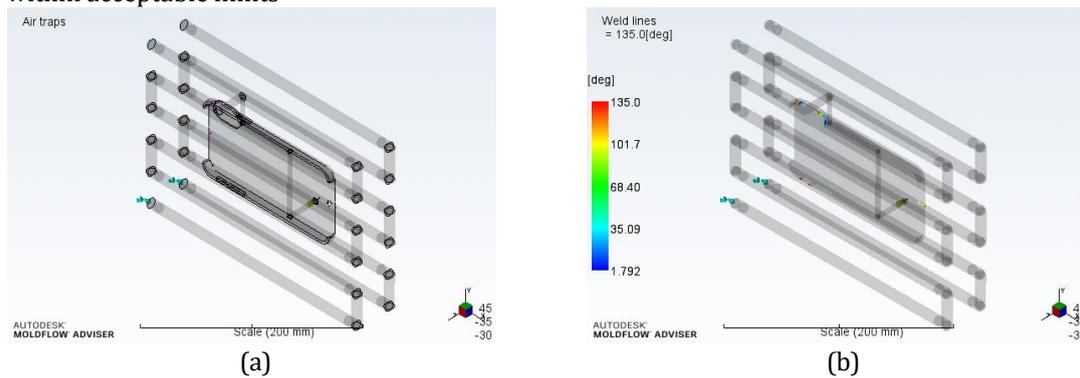

Fig 13 *(a).Air trap and (b). weld line in Runner Design 2*

The longer runner design impacts the quality of the molded part by introducing more variability in flow and pressure distribution. The volumetric shrinkage at ejection is measured at 4.511%, which is slightly higher than that of Runner Design 1, indicating a greater potential for dimensional variations. Warping is also slightly more increased, with a maximum deflection of 0.6284 mm. These issues highlight the challenges posed by the longer and more complex runner path, which requires additional optimization to improve part quality.
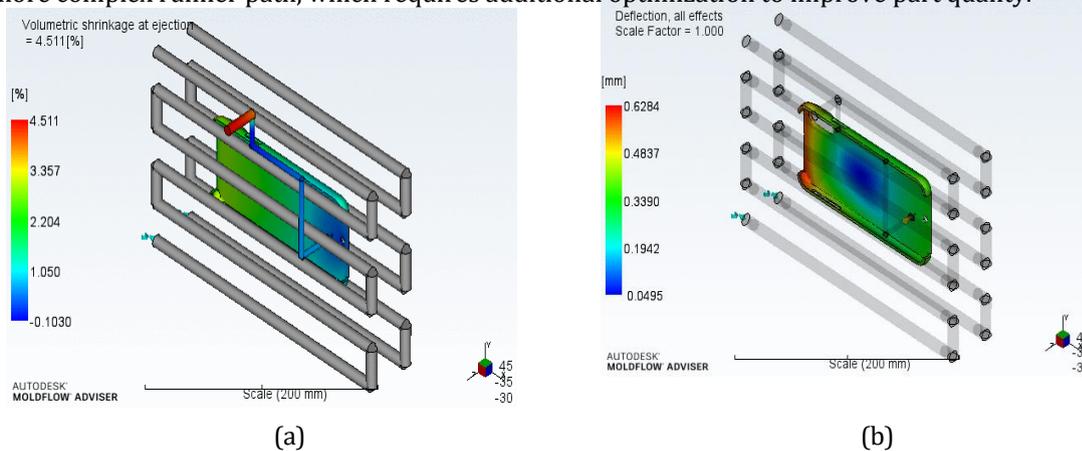

Fig 14: *(a)volumetric shrinkage and (b)sink mark for Runner Design 2*

3.2 Comparison of Analysis

The comparison between Runner Design 1 and Runner Design 2 reveals clear differences in performance. Design 1 has a shorter fill time of 1.362 seconds, compared to 1.407 seconds for Design 2, allowing quicker mold filling. Design 1 also experiences a lower pressure drop of 113.6 MPa, while Design 2 has a higher pressure drop of 147.0 MPa due to its longer, more complex runner path. Both designs achieved a 100% fill confidence.

Regarding weld lines, both designs had a similar angle of 135 degrees. However, Design 2 performed better in sink mark depth with a smaller depth of 0.0171 mm compared to 0.0224 mm in Design 1, indicating better material packing. Design 1 showed lower volume shrinkage at 4.454%, compared to 4.511% in Design 2, demonstrating better dimensional accuracy. Lastly, Design 2 had slightly better part stability with a deflection of 0.6284 mm, while Design 1 had 0.6755 mm.

Table 5 *The comparison between Runner Design 1 and Runner Design 2*

Parameter	Runner Design 1	Runner Design2
Fill time (s)	1.362	1.407
Pressure drop (MPa)	113.6	147.0
Confidence of filling (%)	100	100
Weld line (deg)	135	135
Sink mark depth (mm)	0.0224	0.0171
Volume shringkage (%)	4.454	4.511

Maximum defelction (mm)	0.6755	0.6284

4. Conclusion

This study examined how runner design affects the quality of injection-molded parts using SolidWorks and Autodesk Moldflow simulation software. The results emphasized that careful runner system design can enhance injection molding efficiency and reduce defects like warping, sink marks, and uneven filling. Key factors such as gate location, runner length, and pressure distribution were critical for achieving high-quality, consistent parts. Two runner designs were analyzed based on fill time, pressure drop, material flow, and defects. Runner Design 1 outperformed Design 2 with a shorter fill time, lower pressure drop, and fewer issues like shrinkage and deflection, making it more effective for high-quality and stable parts. Although Design 2 suited complex molds, Design 1 proved more efficient for quality production.

The use of SolidWorks and Moldflow together allowed for easy modeling, simulation, and optimization of the runner system, reducing the need for physical prototypes, saving time, and identifying issues early in the design process. This approach improved molding efficiency and reduced waste. In conclusion, the study highlights the value of design and simulation tools in improving injection molding processes. The findings are relevant for industries such as automotive, electronics, and medical devices. Future research could explore more complex runner designs, different materials, or other process factors to further enhance molding efficiency and part quality.

Acknowledgement

Conflict of Interest

Authors declare that there is no conflict of interests regarding the publication of the paper.

Author Contribution

The author confirms sole responsibility for the following: study conception and design, data collection, analysis and interpretation of results, and manuscript preparation.

References

[1] M. Nidzuwan and M. H. Othman, "The Comparison on Mold Design for Injected Molded Integral Hinges Test Sample Via Autodesk Mold Flow Adviser 2019", doi: 10.30880/rpmme.2021.02.01.022.*Online Magazine Article*

[2] L. Sanchez-castillo, D. Nedelcu, and M. Francisco-marquez, "Redesign of Layout Runner in Rubber Injection Molding for Filling of a Multi-cavity Mold," *Mater. Plast*, vol. 58, no. 3, pp. 121–128, 1964, doi: 10.37358/Mat.Plast.1964.

[3] A. M. Gohn, D. Brown, G. Mendis, S. Forster, N. Rudd, and M. Giles, "Mold inserts for injection molding prototype applications fabricated via material extrusion additive manufacturing," *Addit Manuf*, vol. 51, Mar. 2022, doi: 10.1016/j.addma.2022.102595.

[4] R. D. Párizs and D. Török, "How to use prior knowledge for injection molding in industry 4.0," *Results in Engineering*, vol. 23, Sep. 2024, doi: 10.1016/j.rineng.2024.102667.

[5] C. C. Lin and H. X. Lin, "Injection molding for hollow products with a varying cross-sectional contour by using ice core technology," *Mater Today Commun*, vol. 38, Mar. 2024, doi: 10.1016/j.mtcomm.2024.108258.