OPTIMIZING PLASTIC INJECTION MOLDING PROCESSES FOR PVC: A COMPARATIVE ANALYSIS OF GATE AND RUNNER DESIGNS THROUGH FEA

Tanmay Salke[1], Dr. P.V. Jadhav[2]
[1]PG Student, CAD/CAM, Bharti Vidyapeeth (Deemed to be University) College of Engineering, Pune

[2]Professor, Mechanical Engineering, Bharti Vidyapeeth (Deemed to be University) College of Engineering, Pune
[1]Corresponding Author e-mail: tanmaysalke@gmail.com

ABSTRACT

Plastic injection molding plays a crucial role in meeting the increasing market demand for mass production of plastic components. To optimize the molding process and meet customer expectations in terms of cost and time efficiency, careful consideration of the runner and gate designs is essential. This paper focuses on the analysis and optimization of a plastic injection mold using finite element analysis (FEA). The objective is to determine the most suitable gate and runner combination for an 8-cavity mold by simulating four scenarios: trapezium runner with edge gate, trapezium runner with pinpoint gate, semicircular runner with edge gate, and semicircular runner with pinpoint gate.

The study evaluates various parameters, including fill time, pressure at the end of fill, shear rate, and volumetric shrinkage, using SolidWorks simulation software. The meshing technique is employed to convert the CAD design into a mesh for accurate analysis. The results provide insights into the performance of each runner and gate combination, enabling the identification of the optimal design configuration.

The findings highlight the importance of runner system selection in achieving successful molding, with considerations for factors such as fill time, pressure, and shear stress. The research contributes to the understanding of mold flow analysis and the design optimization of injection molds, ultimately enhancing production efficiency, reducing defects, and minimizing costs.

Keywords: plastic injection molding, runner system, gate design, FEA analysis, mold flow analysis, optimization.

INTRODUCTION

The optimization of plastic injection molding processes is a significant challenge faced by manufacturers and researchers in their quest to produce cost-effective components that meet customer expectations. In today's dynamic market, where demand is rapidly rising, injection molding plays a critical role in achieving mass production of plastic parts. Therefore, it is essential for manufacturers to optimize the cycle time of the molding process to meet market demands efficiently. [1] [4]

Runners provide channels for the molten resin to flow into the mold cavity, while gates, being the thinnest part of the runner system, facilitate quick freeze-off to prevent material from flowing back into the cavity. In multi-cavity molds, the design of the runner system significantly impacts part quality and molding efficiency. Different cross-sectional shapes of runner systems exist, each with its specific applications. Selecting the appropriate runner system shape for a particular product requires careful evaluation of various factors. The ejection of a runner system with rectangular, square, or polygonal shapes presents challenges due to the presence of corners in these cross sections. Incorrect determination of the runner system's shape and dimensions can lead to issues such as pressure drop, incomplete cavity filling, and excessive heat transfer to the mold walls. [3]

Mold flow analysis and the design of an optimal injection mold play a critical role in cost reduction, enhancing filling and cooling parameters, improving packing efficiency, and minimizing mold defects. [7] This paper focuses on conducting a mold flow analysis of a PVC model with an established standard design to identify the optimal gate and runner system for the mold. By analyzing and refining the mold design characteristics, this study aims to improve reliability, simplify the process, and reduce costs. The analysis will involve simulating an 8-cavity mold under four different scenarios: trapezium runner with an edge gate, trapezium runner with a pinpoint gate, semicircular runner with an edge gate, and semicircular runner with a pinpoint gate. The simulations will be performed using SolidWorks software, and critical parameters such as fill time, pressure, weld lines, and shear rate will be examined to evaluate the effectiveness of each scenario.

LITERATURE REVIEW

In a multi cavity mold design, fishbone runner system is most suitable as the wastage is less in runner. It is important to make your design which supports balanced flow to achieve good quality products, lower injection pressure (Chung-Chih Lin, 2022) [4]. The prominent parts of mold are sprue, runner and gate. The function of sprue is to be a channel from nozzle to runner for the molten polymer. The polymer runs through the runners and enters the mold cavity through gate. The cross section of gate is small, (Prachi Kale,2021) [2]. The study compares square and circular runners on various parameters like runner position, stress, strain and deformation (Ravindra K Ugalmugale, 2016) [6]. The study compares four cases using different parameters for runner, gate and sprue for moulding of a toy component (Manmit Salunke, 2015) [5]. The utilization of injection molding analysis software, has been demonstrated to provide valuable information for plastic product and mold design. By optimizing filling behavior and detecting potential defects, the software helps reduce production time and cost, especially for complex parts. (Samiksha S. Borkar, 2021) [7].

PROBLEM STATEMENT

The optimization of plastic injection molding processes for PVC material poses challenges for manufacturers seeking cost-effective production while meeting customer expectations. Selecting suitable gate and runner designs plays a crucial role in achieving successful molding outcomes. However, a lack of comprehensive understanding regarding optimal configurations specifically for PVC material hampers manufacturers from effectively addressing the unique challenges of PVC injection molding.

Existing research highlights the significance of mold flow analysis and design optimization to reduce costs, improve filling and cooling parameters, and minimize defects. Yet, a knowledge gap remains regarding the most effective gate and runner designs for PVC material in injection molding processes.

This study aims to investigate the optimization of plastic injection molding processes for PVC using finite element analysis. The objective is to simulate an 8-cavity mold with four different gate and runner scenarios. By analyzing key parameters such as fill time, pressure, and shear stress, volumetric shrinkage, cooling time, this research aims to provide insights into the optimal designs for PVC material. The findings will empower manufacturers to enhance production efficiency, reduce costs, and achieve high-quality PVC components in injection molding.

OBJECTIVE

1. Create CAD models for fishbone runner system for 8 cavity mold (including sprue, runner, gate) with 4 different scenarios : trapezium runner with an edge gate, trapezium runner with a pinpoint gate, semicircular runner with an edge gate, and semicircular runner with a pinpoint gate.
2. Utilize mold flow analysis to evaluate and optimize the gate and runner system configurations for PVC material in an 8-cavity mold.
3. Simulate four different scenarios, analyze critical parameters such as fill time, pressure, volumetric shrinkage, shear stress, cooling time to evaluate the effectiveness of each gate and runner system configuration in achieving desired molding outcomes for PVC components.
4. Compare and assess the results obtained from the simulations to identify the optimal gate and runner system design for PVC material in injection molding processes.
5. Provide valuable insights and recommendations for manufacturers regarding the selection and optimization of gate and runner designs specifically tailored to PVC material, aiming to improve production efficiency, reduce costs, and achieve high-quality PVC components in the injection molding process.

METHODOLOGY

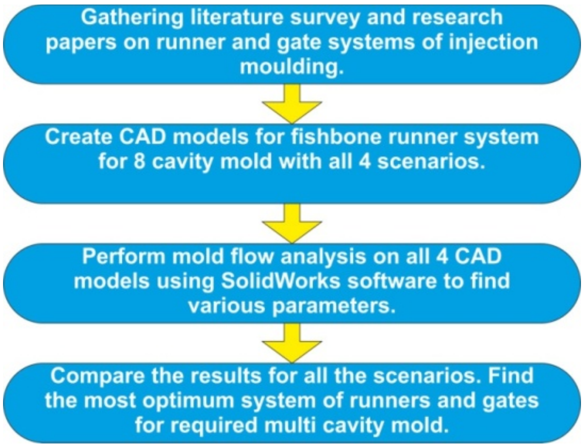

Figure 1. Methodology

CAD DESIGNS

Following is the PVC part for which we are suppose to make multi-cavity mold.

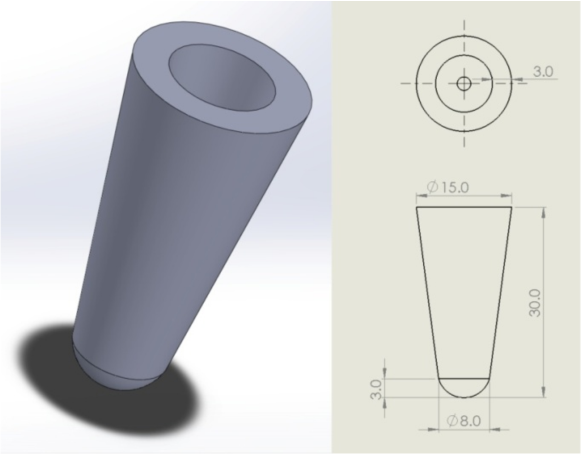

Figure 2. Part design and dimensions

The part is rounded tip conical cap with 3 mm thickness.

Now since the part is not complex and to create multi-cavity mold, our primary aim is to reduce runner wastage. In such scenario it is ideal to use fish bone runner system. [4]

Considering this we will be using fishbone structure for our multi-cavity mold.

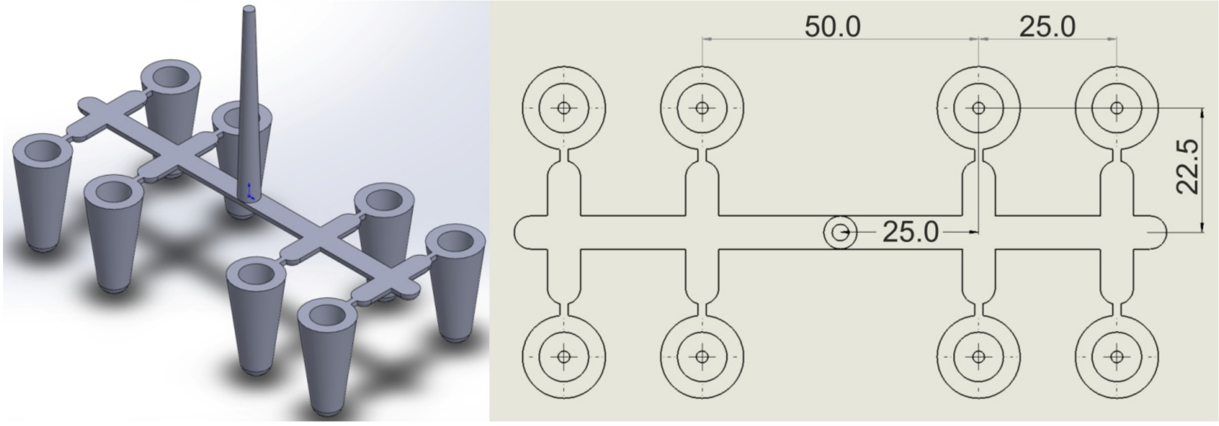

Figure 3. Fish bone runner system with dimensions.

Runner designs

For the 4 scenarios we will be simulating, we will consider 2 types of runners, Trapezium runner and semicircular runner. The cross section area of the runners is as following.

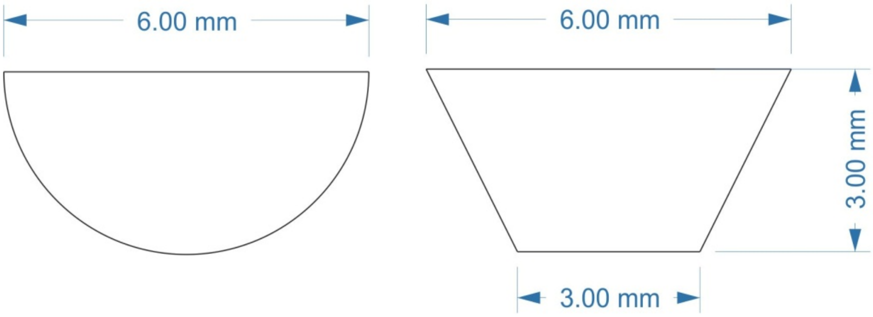

Figure 4. Cross section area of semicircular runner and trapezium runner

The cross sectional area of semicircular runner = 3.14 x 3 x 3 / 2 = 14.13 mm^2

The cross sectional area of trapezium runner = (3 + 6) x 3 / 2 = 13.5 mm^2

Similarly we use two gates whose cross section areas are as shown in figure 5. The edge gate provides flow through larger area, whereas the semicircular pinpoint gate as the name suggests provides smaller area for flow of polymer.

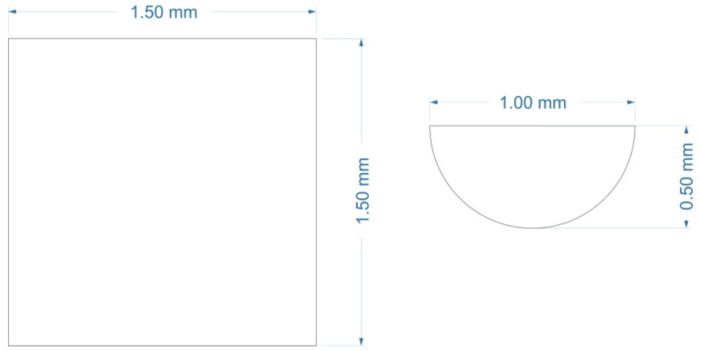

Figure 5. Cross section of edge gate and semicircular pinpoint gate

The cross sectional area of edge gate = 1.5 x 1.5 = 2.25 mm^2

The cross sectional area of semicircular pinpoint gate = 3.14 x 0.5 x 0.5 / 2 = 0.3925 mm^2

Now as per the 4 cases we are going to do mold flow analysis.

Case 1: Trapezium runner with edge gate

Case 2: Trapezium runner with pinpoint gate

Case 3: Semicircular runner with edge gate

Case 4: Semicircular runner with pinpoint gate

A close look of runner, gate and part for the 4 cases is shown in figure 6.

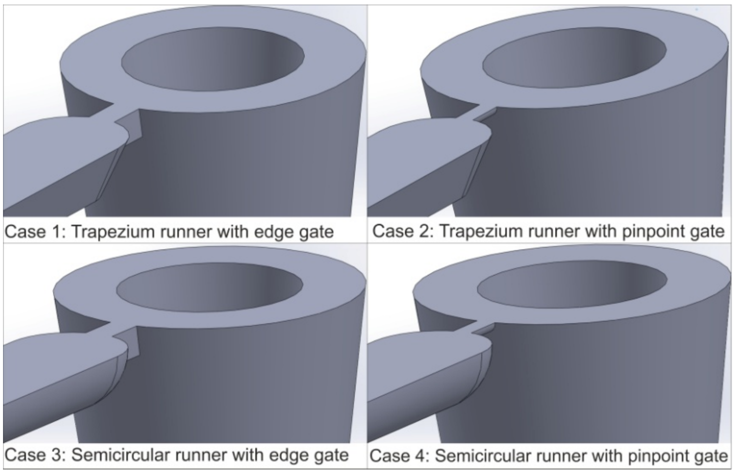
Case 1: Trapezium runner with edge gate
Case 2: Trapezium runner with pinpoint gate
Case 3: Semicircular runner with edge gate
Case 4: Semicircular runner with pinpoint gate

Figure 6. The 4 cases of runner and gate system

MOLD FLOW ANALYSIS

The first step to order to perform mold flow analysis is to convert the CAD file into solid mesh. In the context of CAE (Computer-Aided Engineering) software, a mesh is utilized to carry out calculations and simulations. It is composed of interconnected elements that describe the simulated part. [8] Nodes, in the context of meshing, are points within the mesh where elements connect. These nodes serve as reference points and store important data during the analysis, such as displacement, temperature, or pressure. By defining the behavior and characteristics of the elements at these nodes, the entire mesh accurately represents the simulated part and enables accurate analysis and evaluation of the CAE calculations. Figure 7 shows virtual representation of solid mesh

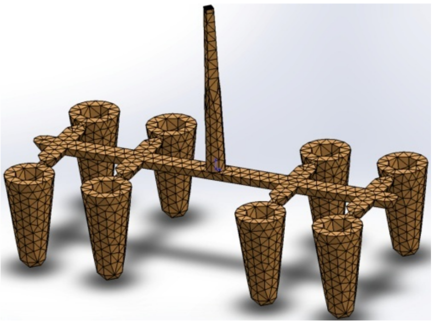

Figure 7. View of Solid mesh

Table 1 shows the number of elements and number of nodes for each of the cases

	Number of elements	Number of nodes
Case 1	210759	145158
Case 2	240961	169987
Case 3	214290	146684
Case 4	235187	165404

Table 1. Number of Nodes and Elements

Injection location is situated on top of the sprue, the pointer diameter we will be taking for simulation is 3mm.

We are taking PVC generic material for simulation of the mold. Table 2 shows material properties for generic PVC material

Material Property	Magnitude
Melt Temperature	$190^{O}C$
Max. Melt Temperature	$220^{O}C$
Min. Melt Temperature	$160^{O}C$
Ejection Temperature	$75^{O}C$
Glass Transition Temperature	$80^{O}C$

Table 2. Material Properties (Generic PVC material)

Considering the material properties we adjust the fill settings accordingly for all the 4 cases. Table 3 shows the settings entered in the software.

Melt Temperature	$190^{O}C$
Mold Temperature	$40^{O}C$
Injection Pressure	100 MPa
Clamp Force	100 Tonne

Table 3. Fill settings

With all the settings done, we simulate the flow. We repeat the process for all four cases and acquire the results for following parameters

1. Fill Time: The duration needed to fill the resin into the mold cavity is referred to as the fill time. Fill time is primarily influenced by material properties, injection pressure limitations, and the characteristics of the gating system used for the resin. [10]
2. Max. Pressure at End of Fill: Max Pressure at the end of fill can be defined as the highest level of pressure achieved during the injection molding process when the polymer flow front moves from regions of high pressure to low pressure. It is similar to the movement of water from higher to lower elevations. As the polymer melt is injected, high pressure builds up at the injection nozzle to overcome the resistance encountered during flow. Subsequently, the pressure gradually decreases along the flow path until it reaches the atmospheric pressure at the polymer flow front, provided that the cavity is adequately vented. [9]
3. Max. Shear Stress at End of Fill: Max shear stress at the end of fill can be defined as the maximum shear force experienced per unit area within the cavity. Shear stress represents the force exerted by the molten material parallel to the plane of the cavity wall, serving as the force that pushes the cavity wall in the direction of flow. It is not directed outward in a normal direction to the wall but rather acts tangentially to the surface.
4. Max. Volumetric Shrinkage: Max. volumetric shrinkage represents the maximum amount of dimensional contraction observed in a plastic material as it undergoes the cooling and solidification phases of the injection molding process.
5. Cooling time: The time taken by the polymer to cool down to the point of ejection is called cooling time. It is important parameter in determining profitability of the mold. [9]

RESULTS

After simulation of Mold Flow we get the following results

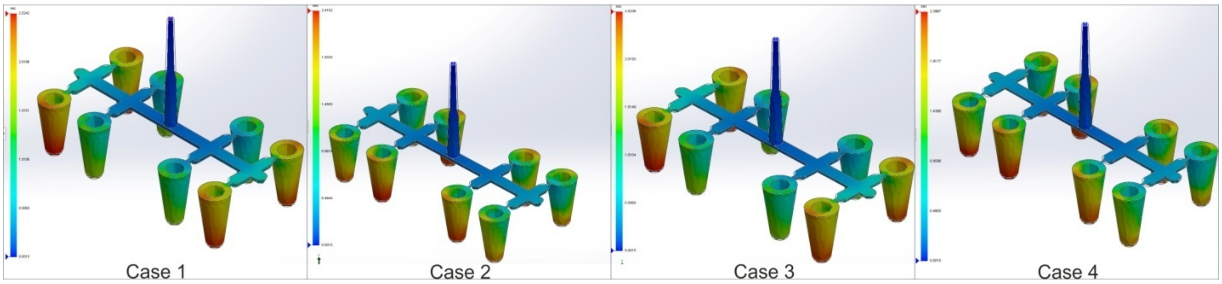

Figure 8. Fill time Analysis

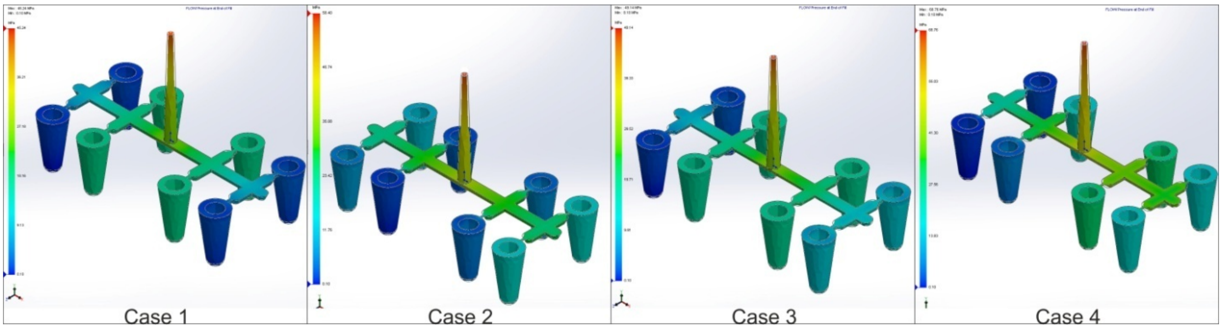

Figure 9. Pressure at End of Fill Analysis

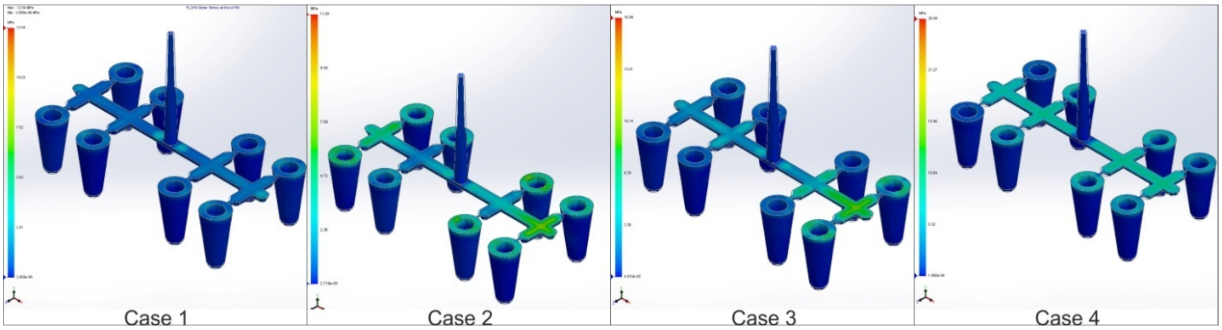

Figure 10. Shear Stress at End of Fill Analysis

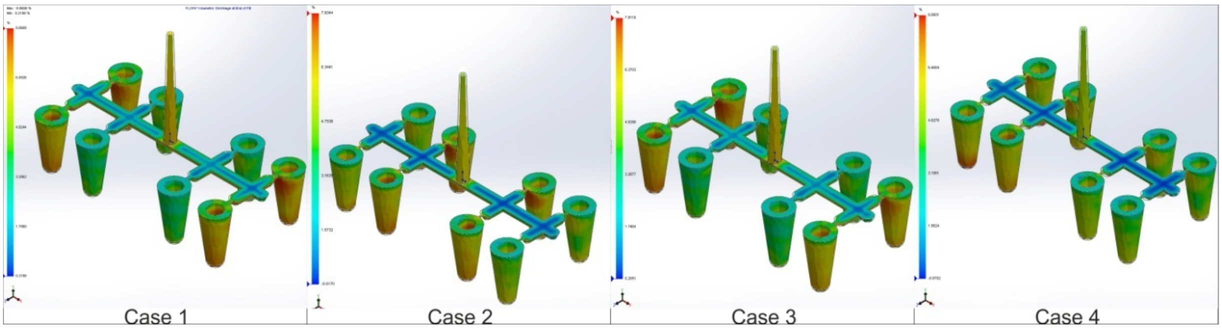

Figure 11. Volumetric Shrinkage Analysis

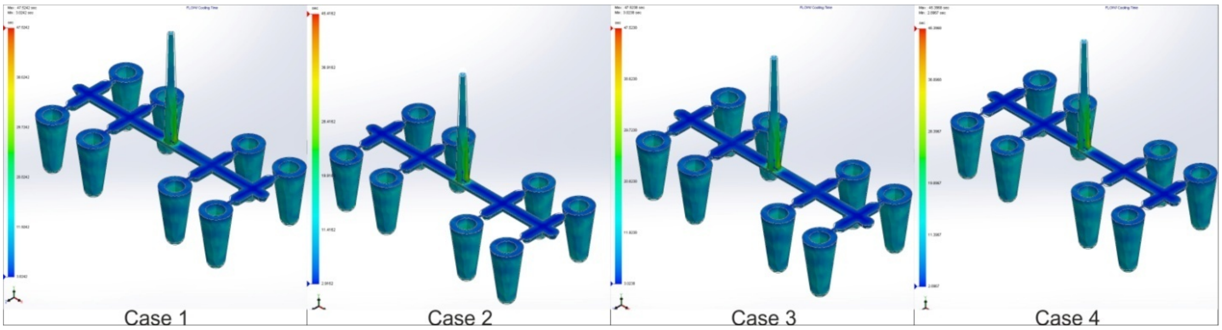

Figure 12. Cooling Time Analysis

Table 4. Shows the acquired results from the analysis

Parameter	**Case 1**	**Case 2**	**Case 3**	**Case 4**
Fill Time	2.5242 sec	2.4162 sec	2.5238 sec	2.3967 sec
Max. Pressure at End of Fill	45.2407 MPa	58.4036 MPa	49.1362 MPa	68.7616 MPa
Max. Shear Stress at End of Fill	12.5379 MPa	11.8104 MPa	16.8920 MPa	26.5893 MPa
Max. Volumetric Shrinkage at End of Fill	8.0608 %	7.9344 %	7.9116 %	8.0931 %
Cooling Time	47.5242 sec	45.4162 sec	47.5238 sec	45.3968 sec

Table 4. Analysis results for four cases

CONCLUSION

To determine the most optimal case among the four scenarios, we need to consider the specific criteria and objectives of our study. The optimal case will depend on the priorities and requirements we have set for our project.

1. Fill Time: Case 4 has the shortest fill time (2.3967 sec), indicating faster and more efficient mold cavity filling compared to the other cases.
2. Max. Pressure at End of Fill: Case 4 has the highest pressure at the end of fill (68.7616 MPa), suggesting better packing of the plastic material and potentially reduced risk of voids or incomplete filling.
3. Max. Shear Stress at End of Fill: Case 4 has the highest shear stress at the end of fill (26.5893 MPa), indicating a higher level of shear-induced stress on the material during filling. Depending on your specific requirements, this could be either desirable or a concern.
4. Max. Volumetric Shrinkage at End of Fill: All cases have relatively similar volumetric shrinkage values, with Case 4 having slightly higher shrinkage (8.0931%). The impact of shrinkage on your component's functionality and quality should be considered.
5. Cooling Time: Cases 2 and 4 have the shortest cooling times (45.4162 sec and 45.3968 sec, respectively), suggesting potentially faster solidification and shorter overall cycle time.

Considering these observations, if a shorter fill time, higher pressure at the end of fill, and shorter cooling time are important factors for your project, then Case 4 could be considered as the most optimal among the four scenarios.

REFERENCES

[1] Prashant K Mulge, Dr. Shashidhar S Kalashetty A Brief Literature Review on Optimization of Plastic Injection Moulding Process Parameters for Various Plastic Materials by using Taguchi’s Technique, International Journal of Engineering Research & Technology (IJERT), ISSN: 2278-0181, Vol. 8 Issue 07, July-2019

[2] Prachi Kale, Ramnarayan Sahu, A literature review on injection moulding process based on runner system and process variables A literature review on injection moulding process based on runner system and process, IOP Conference Series Materials Science and Engineering, January 2021

[3] Mehdi Moayyedian, Romeo M. Marian, New Design Feature of Mold in Injection Molding For Scrap Reduction, Procedia Manufacturing, February 2015

[4] Lin, C.-C.; Wu, T.-C.; Chen, Y.-S.; Yang, B.-Y, A Semi-Analytical Method for Designing a Runner System of a Multi-Cavity Mold for Injection Molding, Polymers 2022, 14, 5442. https://doi.org/10.3390/polym14245442

[5] Manmit Salunke, Rushikesh Kate, Vishwas Lomate, Gajanan Sopal, INJECTION MOLDING METHODS DESIGN, OPTIMIZATION, SIMULATION OF PLASTIC TOY BUILDING BLOCK BY MOLD FLOW ANALYSIS, International Journal of Mechanical Engineering and Technology, Volume 6, Issue 6, June (2015), pp. 33-42

[6] Ravindra K Ugalmugale, Prof. V. B. Shinde, Design and Analysis of Plastic Moulding Die and Runner Shape Optimization by Using Analytical Hierarchy Process, International Journal of Engineering Science and Computing, July 2016, Volume 6 Issue No. 7

[7] Samiksha S. Borkar, Mold Flow Analysis & Injection Mold Tool of Plastic Model, 2021, ISSN 2348-4098

[8] Felipe Marin, Adriano Fagali de Souza, Rodolfo Gabriel Pabst and Carlos Henrique Ahrens, Influences of the mesh in the CAE simulation for plastic injection molding, 2019, ISSN 1678-5169

[9] Jay Shoemaker, Moldflow Design Guide: A Resource for Plastics Engineers (Massachusetts: Hanser Verlag, 2006) page 18,30.

[10] Gurmeet Singh , Sharad Srivastava , Neeraj Kumar Sharma, Optimization of Fill Time in Multi Cavity Plastic Injection Molding Through Simulation, International Journal of Engineering, Management & Sciences (IJEMS)ISSN-2348-3733, Volume-2, Issue-4, April 2015