Hindawi

Research Article

Design of an Automobile Injection Mould Based on Automation Technology

Silin Cao[1] and Guimao Si [2]

[1]*College of Automotive Engineering, Shaanxi College of Communications Technology, Xi'an 710018, China*
[2]*College of Engineering Machinery, Chang'an University, Xi'an 710000, China*

Correspondence should be addressed to Guimao Si; smart@chd.edu.cn

At present, there are frequent occurrences in the industry of manufacturers only having products, but they have no product data, resulting in manufacturers unable to open molds to produce the products themselves and can only buy them at high prices. This article takes the rear case of an automobile generator produced by a company as an example, which aims to show the design of automobile injection molds through automation technology. This paper studies the use of automation technology to achieve the automatic model reconstruction of products, and uses the CAD technology to design injection molds based on the obtained digital models of products. This paper also uses the CAE technology to verify the feasibility and rationality of the designed mold, exploring the optimization method of the injection molding process of the mold. In this paper, a special product pouring system and demolding mechanism are designed according to the product process requirements. The design greatly simplifies the mold structure while fulfilling the product process requirements, which avoids the use of expensive hot runners and complex flip-chip mold structures, reducing the mold manufacturing cost. Secondly, the overall design of the injection mold is carried out, and the external dimensions and other main parameters of the mold are determined. Then, Moldflow software is used to simulate the injection molding process, verifying the feasibility and rationality of the designed mold. The optimization method of the injection molding process is also proposed, and the optimal injection process parameters are determined to ensure that the mold can produce qualified products and ensure high production efficiency. It can be seen from the analysis results that the maximum warpage deformation of the plastic part under all effects is 0.65 mm, which meets the product process requirements.

1. Introduction

The traditional injection mold design mainly relies on the experience of mold engineers, and the manual operation error is large. The mold needs to be debugged and corrected many times before it can be officially put into production. This method has high labor cost, long cycle and large limitations, and it is difficult to meet the requirements of contemporary industry for short development cycle and high production efficiency of plastic products. Compared with the traditional manual drawing method, the application of the CAD technology has the advantages of shorter time consumption, faster speed, and higher precision in assisting the completion of the mold structure design, mold part design, and mold assembly. It can significantly shorten the development cycle of new molds and improve the working stability of molds. In addition, the application of the CAE technology in the development of injection molds can detect design defects in advance and correct the design plan in time before the mold is opened. Meanwhile, the application of the CAE technology can realize the exploration of the best injection molding process parameters of the mold, which can significantly improve the current situation of frequent mold repairs and excessive mold trial workload caused by the use of traditional methods in the industry. The traditional method of mold design and process debugging is a time-consuming and wasteful trial-and-error process. The research in this paper gives full play to the advantages of

automation technology. Especially in the case where the customer only provides the actual plastic parts without the drawings, mold development is difficult and time-consuming, and the use of automated means to assist the development of injection molds has a significant role in reducing the difficulty of mold development and shortening the development cycle. At the same time, it not only plays an important role in improving the current situation that the level of molds in the industry relies heavily on the personal experience of designers and frequent mold repairs, but also reduces labor and material resource costs, improving mold performance and extending mold life.

The research on automobile injection molds has been ongoing all the time. Huang et al. carried out numerical research on the injection molding of carbon fiber composite automobile wheels with complex shapes [1]. Seo et al. focused on the micro-battery injection molding process to develop lightweight automotive handrails made of thermoplastic polypropylene and chemical blowing agents [2]. Guo et al. proposed a combined in-mold decoration and microcellular foam injection molding process (IMD/Mucell) to improve the surface quality of foamed parts [3]. Park et al. presented the design of an injection mold cooling system for the production of plastic automotive door modules [4]. Cho et al. designed a 3D-printed insert for injection tooling to reduce weld line defects in automotive crash pads with a high-standard appearance [5]. Guo aimed to study the effect of molding parameters on the surface roughness of injection-molded polypropylene parts [6]. Contreras et al. achieved a balance between the selected properties of expanded polypropylene (PP) by injection molding [7]. Wang et al. developed a novel rapid mold heating and cooling method featuring electrical heating and annular cooling [8]. Xiao and Huang developed a rapid thermal cycle molding (RTCM) technique with electric heating and water cooling [9]. Kim and Jeon fabricated a mold to produce specimens using long fiber thermoplastics [10]. This type of research still requires a certain amount of labor, and thus automation needs to be used to improve it.

Many scholars have studied automation technology. Song and Zhao studied the development trend of mechanical automation technology, analyzed the design of the program reasoner, and proposed several application strategies of mechanical automation technology [11]. Sun et al. analyzed and expounded the construction problems and practical significance of the military-civilian integration emergency logistics system based on automation technology. He proposed the strategy of constructing the military-civilian integration emergency logistics system with automation technology under the new system [12]. This kind of research lacks data explanation, and the conclusion is still open to question. Therefore, combining the above two researches and taking the automobile as the research object, this article will study the design of an automobile injection mold based on automation technology.

This paper compares the degree of warpage deformation of the product under the same injection molding process conditions under the three design schemes to verify whether the design structure is reasonable. The results show that under the same injection molding process conditions, the maximum warpage deformation of the single-point pouring product is 1.23 mm. The maximum warpage deformation of the two-point pouring product is 0.85 mm. The maximum warpage deformation of the three-point pouring product is 1.17 mm. From the results and analysis, the following aspects can be seen: (1) The two-point pouring design among the three schemes is relatively reasonable; (2) The positions where the products warp is exactly the same in all schemes, and thus it can be tried to separate the causes of warpage in order to further explore the causes of different warpage values.

2. Design of the Automobile Injection Mold

The range of automation technologies is extensive and we have used both Computer-Aided Design (CAD) and Computer-Aided Engineering (CAE) technologies in this design; this paper has completed the UG-based motor rear shell mold CAD and the Moldflow-based injection mold CAE, and finally made the automobile. The design of the injection mold can meet the needs of the research.

2.1. UG-Based Motor Rear Shell Mold CAD. Plastic products have higher requirements for injection molds, and the traditional pure manual design method can no longer meet the high-standard product process requirements. Under the influence of the rapid development of contemporary computer technology, the CAD technology has been applied in the machinery industry, and the design method of injection molds has also developed from traditional manual drawing to the use of CAD technology for mold design. This method has become the most effective way to solve the above problems. Compared with traditional methods, injection mold CAD technology has all-round advantages in improving production efficiency, improving mold quality, and reducing production costs [13, 14].

CAD, computer-aided design, not only has been limited to calculation and drawing, but also has gradually developed into a system that closely combines computer technology and product design technology. It is a product design process that adopts a man-machine combination, which utilizes the strengths of the computer and the designer, respectively, making the system obtain more ideal results [15].

UG (Unigraphics NX) is one of the most widely used software in the injection mold industry. UG has successively developed modules such as motion simulation, electromechanical design concept, ship design, injection mold guidance, and vehicle manufacturing automation on the basis of inheriting the functions of traditional software such as wireframe, surface, and solid. Using parametric design, it has very powerful two-dimensional and three-dimensional modeling functions, which can fully meet the needs of the designers.

Due to the variety of plastic products, complex features, and different requirements, it is essential to conduct a

comprehensive analysis and research on plastic parts before mold design. The analysis contents mainly include product draft detection, product volume, product thickness measurement, and product process requirements. Since the molding methods and the characteristics of the polymer raw materials are different, the mold design should be carried out on the basis of fully considering the characteristics of the selected raw materials. According to the product model analysis and the existing product originals, the following can be concluded: The draft analysis of the model is in good condition, with no negative angle draft feature, and there is no need to design an inclined ejector mechanism. The average wall thickness is 1.5 mm, and the maximum wall thickness is 2.95 mm. Although the average thickness is the best design in principle, the maximum wall thickness is still within the allowable range of product thickness (3.5 mm), indicating that the product model meets the mold opening requirements. Product-related process requirements include the following aspects: There is no obvious casting trace on the front of the product. The warpage deformation is controlled below 0.7 mm. The production raw material is nylon 66 containing 30% glass fiber.

According to the number of injection mold cavities, injection molds are divided into two types: single-cavity molds and multi-cavity molds. Among them, plastic parts produced by the single-cavity mold have high precision. Its process parameters are easy to control, and the mold structure is relatively simple. The manufacturing cost is also low. Multi-cavity mold plastic parts have a high forming rate but low precision, which is difficult to control. It has a complex mold structure and high manufacturing cost. Since the product is an automobile assembly with high precision requirements, a single-cavity design scheme is selected.

The product uses nylon 66 containing 30% glass fiber as the raw material, and the shrinkage rate of the raw material is 0.65% by consulting the relevant manual. The three-dimensional model of the product is manually proportionally scaled with reference to this data. The product starts from the parting surface. The height of the upper part of the mold core is 30 mm, and the height of the lower part of the mold part is 10 mm. In this paper, the overall thickness of the upper die core is 55 mm, but considering the distribution of the cooling water path of the die core, the thickness of the lower die core is increased to 25 mm, and the overall thickness is 35 mm. The mold is opened after filling holes, building blocks, extracting surfaces, dividing blocks, and designing inserts. The mold core design method used in this paper not only takes into account the cost of the mold and realizes its function, but also ensures the rigidity and service life of the mold core.

Due to the consideration of mold life and processing and manufacturing, the use of inserts in modern injection molds is increasing. In this paper, inserts are used in places where the mold is prone to wear, places where the heat dissipation is not good, and places where it is difficult to machine at one time. The insert design has the following advantages:

Good for exhaust. Exhaust is one of the keys to the success of mold design, and exhaust performance is related to whether the quality of the product can meet the requirements. Design inserts in the mold cavity where air is likely to accumulate, and the matching gap between the inserts and the mold core can be used to achieve exhaust.

Extend mold life. Under normal circumstances, inserts are designed at the easily damaged parts of the mold. Once the inserts are damaged, the mold can continue to be used as long as the new inserts are replaced, thus greatly extending the life of the mold.

The template must have sufficient strength and rigidity to carry the movement of the guide post and the thimble during the molding process of the plastic part. In this design, the thickness of the fixed template is 80 mm, and the thickness of the movable template is 75 mm. However, considering that a considerable number of thimbles will pass through the movable template, the thickness is specially designed to be 80 mm to ensure its working life without affecting its function. Since this design adopts the mode of cooperation between the mold core and the template, the right angle part of the fixed and movable template and the mold core is designed by the clear angle method. The diameter of the clear angle is 30 mm. In order to minimize the phenomenon that the mold parting surface runs over the cape due to the loose mold clamping caused by various unexpected factors, a gap of 1 mm is designed between the fixed and movable templates in this design. That is to say, in the state of completion of cooperation, the thickness of the upper and lower mold cores is 0.5 mm higher than that of the corresponding fixed and movable mold plates, which can better realize the clamping and closing of the mold.

The gating system generally consists of a main runner, a branch runner, and a gate. When designing the gating system, focusing on the characteristics of the raw materials of the product and related process requirements, the first consideration should be to make the molten material fill the cavity as quickly as possible to reduce heat and pressure loss. Secondly, the volume of condensate in the flow channel should be minimized to reduce the waste of raw materials. Finally, it should be tried to make the gate traces left on the product small and easy to remove and other factors to design.

At present, the most common way of thermoplastic mold sprue is to insert a detachable nozzle into the template and fix it with a sprue sleeve. Combined with the injection molding machine data provided by the manufacturer, the design results are as following aspects: The inlet diameter of the main flow channel is 3.5 mm. The taper of the flow channel is 1 degree, and the length is 75 mm. Among them, the first-stage cold material hole is designed according to the industry's design practice, and the initial diameter is twice the maximum diameter of the main flow channel. The depth is 10 mm. In this paper, the design adopts two-point pouring. This design method will inevitably produce multi-stage runners. This mold is designed with a U-shaped cross section runner, as shown in Figure 1.
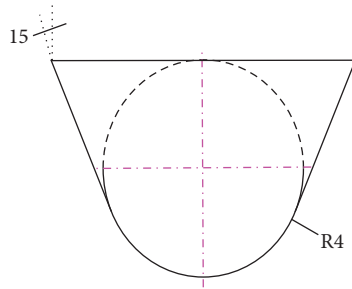
FIGURE 1: Schematic diagram of the U-shaped runner design.

FIGURE 2: Schematic diagram of a part of the gating system.

The design of the secondary runner is a combination of the template and the mold core for the fixed mold in this design. The secondary runner adopts the design method of the main runner, and the shape is a cylindrical with a tapered angle. At the same time, in order to minimize the problems caused by machining accuracy, the secondary runner is divided into two sections from the mating surface of the fixed template and the mold core. Finally, in order to improve the automation level of mold production and reduce labor costs, this design adopts to increase the runner of the beam mouth to realize the automation of the condensate shedding off of the runner of the mold as shown in Figure 2.

After calculation and combining with the characteristics of the mold, it is finally determined that the diameter of the large end of the fixed template section of the secondary runner is 5.5 mm, and the taper is 1 degree. The diameter of the big end of the upper die core segment is 3.8 mm, and the taper is 1 degree. The runner part of the beam mouth is designed and calculated by the approximate point gate method. The final height of the beam mouth is 3.5 mm, and the minimum diameter is 2 mm. The beam mouth runner is directly connected to the secondary cold material cavity and the product gate.

There are many types of injection mold gates, which will not be described in this article. Combined with the original intention of this design and the characteristics of different types of gates, this article uses side gates for pouring.

The characteristics of the side gate include that the cross-sectional area of the gate is small. The distance is short and the melt is easy to solidify. The high degree of mold automation can significantly improve production efficiency. This type of gate is often used in the design of three-plate molds, and the runner at the beam mouth will be automatically broken when the three-plate mold is opened, such that the runner condensate can be smoothly released from different parting surfaces.

This design gate adopts a standard rectangular section. After calculation, it is finally determined that the side gate size is 1.5 mm long and the section shape is a standard rectangular section side gate of 3×1.5 mm.

Due to the special method adopted in this design, there is a secondary cold slug cavity. In order to realize the automatic ejection of the cold slug cavity and the smooth falling off, a corresponding ejection mechanism should be designed at the joint between the gate and the cold slug cavity. The design method of the cold slug cavity is similar to that of the first-stage cold slug cavity, and its special point is that the
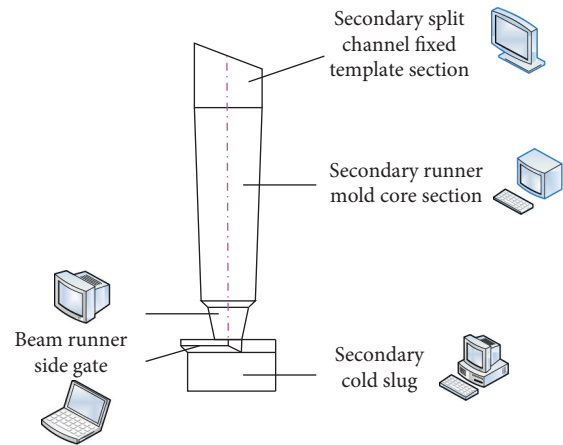
draft angle of the cold slug cavity is increased in order to eject the condensate in the cold slug cavity smoothly. The draft angle is increased from the commonly used 1° to 4°, and the ejection of the cold material cavity can be well accomplished under the action of the specially designed ejection system, as shown in Figure 3.

A system that can control the temperature of the mold within the required temperature range of the raw material of the plastic part and can rapidly cool the melt is called a cooling system. The cooling system not only has a significant impact on product production efficiency, production cost, and economic benefits, but also determines the quality of product production. The cooling system design mainly evaluates two aspects: One is whether it can effectively reduce and control the mold temperature. The other is whether it can achieve uniform cooling of the whole body of the product to reduce the warpage caused by uneven cooling.

In actual production, the commonly used cooling media for injection molds are mainly water and oil. Due to the characteristics of water's large specific heat capacity, low cost, stable temperature, and easy control, this paper uses natural water as the cooling medium to design the cooling system. The designed cooling water channel has a diameter of 8 mm and a total length of about 800 mm, and its direction is consistent with the shape of the mold. The average distance between the centerline of the cooling water channel and the surface of the cavity is 2.0 times the diameter of the water channel. Adding a rubber gasket with a thickness of 2.5 mm, an inner diameter of 8 mm, and an outer diameter of 16 mm to the junction of the mold core waterway and the movable and fixed template waterways, cooling water leakage can be prevented.

The mold is specially designed with an automatic gate cutting mechanism according to the product process requirements. This mechanism can realize automatic gate cutting inside the mold through multiple ejection actions set on the injection molding machine before the mold is opened. The advantages of this design include that there are no casting marks on the surface of the product, and the degree of mold automation is high, which can greatly reduce labor costs and achieve the purpose of no casting marks on the product surface and a high degree of automation. This design is based on the three-plate mold. The side gate cut-off
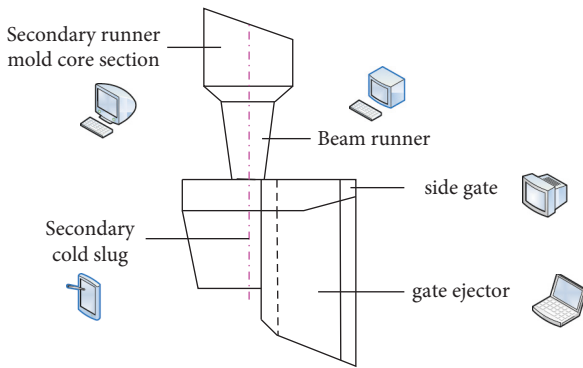
FIGURE 3: Schematic diagram of the runner, gate, and ejector system.

and ejection mechanism are designed separately, and two ejector plates are added to realize fixed-stroke ejection and secondary ejection.

Mold exhaust means that some gases originally existing in the cavity, such as volatiles, moisture, and air, must be expelled from the cavity during the process of injecting the molten material into the cavity in order to prevent the plastic parts from being unfilled due to the accumulation of gas in a certain position of the cavity and unable to be discharged, which will cause the plastic parts to be scrapped. The exhaust system designed in this paper adopts the design of inserts and thimbles at the position where the exhaust is required, and utilizes the tiny gaps between the inserts, the thimbles, and the mold core to achieve exhaust. At the same time, the mold core is processed with special exhaust steel for injection molds to achieve good mold exhaust and reduce the production defect rate of plastic parts.

The demoulding action is described as follows:

The injection molding machine continues to perform the mold opening action. The nylon shutter is forcibly pulled open under the action of the limit pull rod, and the parting surface between the movable and fixed mold plates is opened.

The injection molding machine pushes the bottom plate of the ejector plate. The common ejector pin and the gate ejector pin move at the same time, and the common ejector pin completely ejects the product. After the gate thimble completes the cutting in the gate, due to the unique design of the cold slug well, the cold slug cavity will be ejected out of the step under the action of the fixed-distance stroke. At this time, under the ejection action of the specially designed gate ejector, the secondary cold material hole is smoothly ejected, and the product and the condensate are automatically dropped.

2.2. Moldflow-Based Injection Mold CAE. With the rapid application of computer technology in the industry, mold CAE has also been rapidly promoted. Practice has proved that the application of injection mold CAE can accurately predict the filling, pressure holding, and warpage of the

molten material before the mold is manufactured, which is convenient for designers to find problems in advance and modify them in time.

Moldflow is a CAE software for plastic products. It has strict requirements on mesh type and meshing quality. Three mesh types are provided for different types of plastic parts: neutral surface mesh, dual-layer mesh, and 3D mesh. Meshing principles refer that the number, density, and aspect ratio of meshes (as shown in Figures 4(a) and 4(b)) all affect the analysis results. The ideal mesh state is that all meshes are equilateral triangles, but it is difficult to achieve in practical engineering. Therefore, for small plastic parts, the mesh aspect ratio should be less than 6, and the mesh aspect ratio of large- and medium-sized plastic parts should be less than 12. There should be no free edges, no intersecting elements, and overlapping elements. For flow simulation analysis, the grid matching rate is required to be no less than 85%. For warping analysis, the grid matching rate is required to be no less than 90%.

For ordinary plastic parts, the analysis process of Moldflow is shown in Figure 5.

Moldflow's thermoplastic injection mold flow analysis is mainly composed of three modules: flow analysis, cooling analysis, and warpage analysis. Each module contains several sub-analysis items, which can effectively help engineers predict the advantages and disadvantages of multiple design schemes. When providing scientific predictions for mold design engineers, it effectively saves time and greatly reduces the risk of mold opening.

Because Moldflow software is a professional mold flow analysis software, the modeling function is not powerful. Therefore, before performing mold CAE, UG is used to export the stl format file that Moldflow can recognize, and it will be imported for finite element mesh division.

Since the plastic parts are thin-walled, cylindrical, and deep-cavity products, a double-layer mesh is used. In the finite element method, the quality of the model mesh is critical to the analysis results. Moldflow provides a variety of methods for mesh quality verification and adjustment, such as mesh connectivity, aspect ratio and merging nodes, and filling voids. After adjusting the aspect ratio of the mesh, adjusting the element orientation, and removing the free edge, a product mesh model that meets the requirements of the mold flow analysis is obtained.

The raw material for plastic parts production is PA66 + 30% GF. Before the analysis, the plastic parts' raw materials should be selected in the Moldflow raw material library. Moldflow has its own raw material library and recommended process for corresponding raw materials, as shown in Table 1.

The position of the gate has a huge impact on the molding quality of the plastic part, which is directly related to the flow of the melt in the mold cavity, and it will have a direct impact on the fiber orientation of the polymer and the warpage of the plastic part after molding. Therefore, it is very important to choose the appropriate gate location.

Before selecting the design of the gating system, it is necessary to predict the optimal gate position for the molding process of the plastic part. Since the raw material of
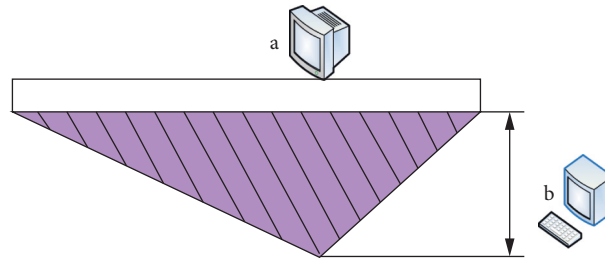
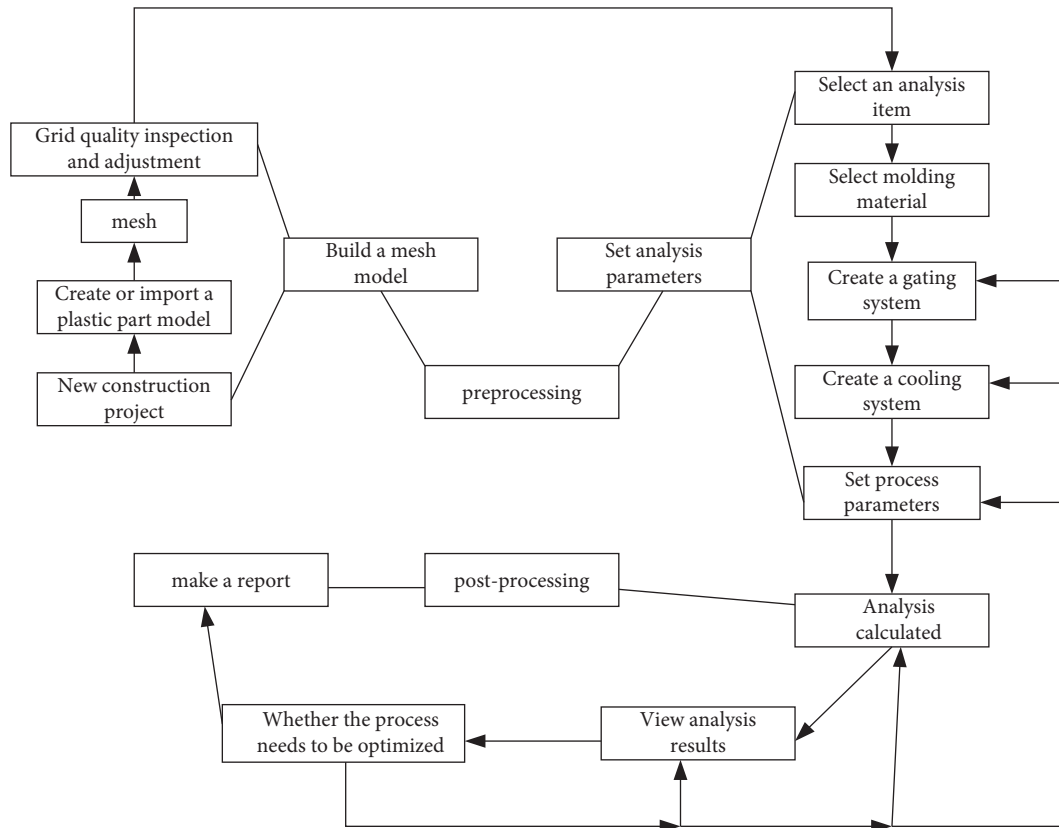FIGURE 4: Schematic diagram of the grid aspect ratio.

FIGURE 5: Moldflow analysis process.

TABLE 1: Recommended process for raw materials.

Mold temperature range (recommended)		Melt temperature range (recommended)	
Max (℃)	100	Max (℃)	290
Min (℃)	50	Min (℃)	270
Mold surface temperature (℃)	75	Ejection temperature	180
Melt temperature	280	Maximum shear stress (MPa)	0.5
Absolute maximum melt temperature	300	Maximum shear rate 1 (s)	60000

plastic parts contains glass fibers, and the position and number of gates have a decisive influence on the orientation of the material fibers, a multi-plan comparison method is adopted to verify whether the previous design is reasonable. Moldflow software is used to determine the optimal gate position, and three schemes of one-point pouring, two-point pouring, and three-point pouring are selected for analysis. It can be seen from the analysis results that the best gate position recommended for single-point pouring is in the center of the plastic part, and the two-point pouring recommended the best gate position be in the middle of the heat dissipation holes on both sides of the plastic part. For three-point pouring, it is recommended that the best gate position be evenly distributed every 120° on the circumferential surface of the plastic part.

The product requires no casting marks on the surface. If single-point pouring or three-point pouring is selected, in order to complete the product process requirements, the

mold must be designed by the flip-chip method. Therefore, in Moldflow, the flow channel is created directly according to the flip-chip design. The method of creating a gating system in Moldflow is: (1) to create or import the centerlines of all runners and gates in the gating system; (2) to specify the corresponding attributes according to the classification of the main channel, runner, and gate in the gating system; (3) to specify the length of the cylindrical element mesh according to the cross-sectional area of each part of the runner.

The corresponding gating system is created based on the analysis results of the gate positions of the one-point pouring (scheme 1), two-point pouring (scheme 2), and three-point pouring (scheme 3) schemes.

The modeling function of Moldflow is not powerful, in order to explore the working conditions of the mold to the greatest extent and verify the cooling effect of the water channel. This paper adopts the method of "extracting virtual curve" in UG to obtain the axis of the designed cooling water circuit. All lines and corresponding coordinate systems are imported into Moldflow in igs format to create a cooling water channel mesh model. The method of creating a cooling system in Moldflow is as follows: (1) Import the centerline of the cooling water circuit, and specify the properties and mesh diameter of the imported centerline; (2) Specify the length of the cylindrical element grid according to the design results, and generate the cylindrical element grid of the cooling system; (3) Set the water inlet and outlet of the water circuit.

In order to achieve the comparison of the three gating system design schemes above, in this calculation, all the injection molding process parameters use the default values of Moldflow under the conditions of ensuring that all schemes do not have short shots, insufficient pressure holding time, and insufficient cooling time (As shown in Table 2). Since all schemes use the exact same cooling system, Moldflow analysis shows that the very small difference caused by cooling is negligible, and thus the cooling analysis will not be described in this section.

In this paper, the relevant technical parameters of the gating system are solved by extension theory, which makes a general understanding of the research process of injection molds.

Matter-element is a set of triples consisting of thing ζ, feature κ, and magnitude λ of the thing about that feature as

$$\Gamma = (\zeta, \kappa, \lambda). \tag{1}$$

The one-dimensional event element consists of a triplet consisting of action Δ, the feature name of the action χ, and the magnitude σ about feature χ

$$\Theta = (\Delta, \chi, \sigma). \tag{2}$$

The one-dimensional relation element is a triple composed of relation symbol Ξ, relation feature name ε, and corresponding relation feature value μ:

$$\Lambda = (\Xi, \varepsilon, \mu). \tag{3}$$

TABLE 2: Injection molding process parameters.

Index	Parameter	Index	Parameter
Filling time	Defaults	Holding pressure (%)	80
Injection pressure	Defaults	Mold opening time (s)	5
Clamping force	Defaults	Cooling time	Defaults
Holding time (s)	10	Coolant temperature (°C)	25

Assuming that the universe of discourse is represented by Φ, any element of Φ is represented by γ, and $\psi(\gamma)$ is a mapping from Φ to Γ. An extension set in the universe of discourse is represented as

$$\Omega = \{(\gamma, \delta) | \delta = \psi(\gamma), \quad \gamma \in \Phi, \delta \in \Gamma\}. \tag{4}$$

$\psi(\gamma)$ represents the correlation function of set Ω, and δ represents the correlation degree of γ with respect to set Ω.

The positive, negative, and zero bounds of the set are shown as formulas (5)–(7):

$$\Omega^+ = \{(\gamma) | \psi(\gamma) \geq 0, \quad \gamma \in \Phi\}, \tag{5}$$

$$\Omega^- = \{(\gamma) | \psi(\gamma) \leq 0, \quad \gamma \in \Phi\}, \tag{6}$$

$$\Omega^0 = \{(\gamma) | \psi(\gamma) = 0, \quad \gamma \in \Phi\}. \tag{7}$$

Distance represents the distance between a point and an interval. Assuming that Ω is any point in $(-\infty, +\infty)$, and $\omega_0 = \langle \varpi, \rho \rangle$ is any interval in $(-\infty, +\infty)$, the distance from interval ω_0 is as

$$\xi(\Omega, \omega_0) = \left| \Omega - \frac{\varpi + \rho}{2} \right| - \frac{1}{2}(\rho - \varpi). \tag{8}$$

When the optimal point is $\Omega_0 \in (\varpi, \varpi + \rho/2)$, the left distance of any point Ω about Ω_0 and interval ω_0 is as

$$\xi(\Omega, \Omega_0, \omega_0) = \begin{cases} \varpi - \Omega, & \Omega \leq \varpi, \\ \Omega - \rho, & \Omega \geq \Omega_0, \\ \dfrac{\rho - \Omega_0}{\varpi - \Omega_0}(\Omega - \varpi), & \Omega \in \langle \varpi, \Omega_0 \rangle. \end{cases} \tag{9}$$

When the optimal point is $\Omega_0 \in (\varpi, \varpi + \rho/2)$, the right distance of any point Ω with respect to Ω_0 and interval ω_0 is as

$$\xi(\Omega, \Omega_0, \omega_0) = \begin{cases} \varpi - \Omega, & \Omega \leq \Omega_0, \\ \Omega - \rho, & \Omega \geq \rho, \\ \dfrac{\varpi - \Omega_0}{\rho - \Omega_0}(\rho - \Omega), & \Omega \in \langle \Omega_0, \rho \rangle. \end{cases} \tag{10}$$

Assuming that there are two intervals on the number line, $\omega_0 = \langle \varpi, \rho \rangle$, $\omega = \langle \forall, \mathbb{C} \rangle$, and $\omega_0 \subset \omega$; Ω are any points.

The position value of point Ω about the interval set formed by ω_0 and ω is as

$$\partial(\Omega, \Omega_0, \omega_0) = \begin{cases} \xi(\Omega, \omega) - \xi(\Omega, \omega_0), & \Omega \notin \omega_0, \\ -1, & \Omega \in \omega_0. \end{cases} \quad (11)$$

When the location of point Ω is different, its associated function (Ω) is also different:

$$(\Omega) = \frac{\xi(\Omega, \Omega_0, \omega_0)}{\partial(\Omega, \Omega_0, \omega)}. \quad (12)$$

The mold design experience formula is often used in the determination of mold structure size and the check of related design parameters. For example, when designing the gate position of large plastic parts, it is necessary to check the maximum flow distance ratio of the plastic melt, i.e., check the flow ratio. The check formula is as:

$$\beth = \sum_{\mathscr{H}=1}^{\mathscr{G}} \frac{\mathfrak{H}_{\mathscr{H}}}{\hbar_{\mathscr{H}}} \leq \beth_{\max}. \quad (13)$$

In formula (13), $\beth$ is the flow ratio, and $\beth_{\max}$ is the allowable flow ratio range of the plastic. $\mathfrak{H}_{\mathscr{H}}$ is the length of each segment of the melt flow path (mm); $\hbar_{\mathscr{H}}$ is the cavity thickness (mm) corresponding to each segment of the flow path, and $\mathscr{L}$ is the total number of flow paths.

The matter-element expression using quaternary matter-element to represent the knowledge of twisting design can be expressed as

$$\mathfrak{F} = \begin{bmatrix} \wp & \varkappa_1 & \mathscr{L}_1 & \langle \beth_1^+, \mathscr{F}_1^- \rangle \\ & \varkappa_2 & \mathscr{L}_2 & \langle \beth_2^+, \mathscr{F}_2^- \rangle \\ & \cdots & \cdots & \cdots \\ & \varkappa_g & \mathscr{L}_g & \langle \beth_g^+, \mathscr{F}_g^- \rangle \end{bmatrix}. \quad (14)$$

In formula (14), $\wp$ is the gate; $\varkappa$ is the feature of the gate; and $\mathscr{L}$ is the feature value.

According to the matter-element knowledge expression of each component of the gating system, the matter-element model of the complete gating system structure can be established as

$$\mathbb{Q} = \begin{Bmatrix} \mathscr{R} \\ \mho \\ \aleph \\ \beth \end{Bmatrix}. \quad (15)$$

In formula (15), $\mathscr{R}$, $\mho$, $\aleph$, and $\beth$ are the matter elements corresponding to the main channel, the runner, the gate, and the cold material hole, respectively.

Replacing the nodes of the antecedent and the consequent in the production rule with the improved quaternary matter-element, the obtained extension matter-element-based design knowledge rule can be expressed as

$$\text{if } \beth_1 \text{ then } \beth_2. \quad (16)$$

The structure of the extension matter-element table is shown in Table 3.

TABLE 3: Extension matter-element table structure.

Column name	Type of data	Length	Sky
Matter number	Int	4	No
Thing element name	$n$char	30	No
Matter feature name	$n$char	30	No
Matter-element eigenvalues	Float	10	No
Improved engineering parameters	$n$char	30	No
Deteriorated engineering parameters	$n$char	30	No

When the constraint eigenvalues of the gating system are in the numerical interval, the associated function is as

$$(\Omega) = \begin{cases} \dfrac{\xi(\Omega, \Omega_0, \omega_0)}{\partial(\Omega, \Omega_0, \omega)}, & \partial(\Omega, \Omega_0, \omega) \neq 0, \\[2mm] -\dfrac{\xi(\Omega, \Omega_0, \omega_0)}{|\rho - \varpi|} + 1, & \Omega \in \omega_0; \partial(\Omega, \Omega_0, \omega) = 0, \\[2mm] 0 & \Omega \notin \omega_0; \Omega \in \omega; \partial(\Omega, \Omega_0, \omega) = 0, \\[2mm] -\dfrac{\xi(\Omega, \Omega_0, \omega_0)}{|\rho - \varpi|} - 1 & \Omega \notin \omega_0; \Omega \notin \omega; \partial(\Omega, \Omega_0, \omega) = 0. \end{cases} \quad (17)$$

Assuming that the material of a large plastic product is PP, the injection pressure during molding is 120 MPa, and the process ratio interval of the material is (0, 300), the correlation function of the plastic part mold gating system scheme on the process ratio condition is as

$$(\Omega) = \begin{cases} -\dfrac{300 - \Omega}{300} + 1, & \Omega \in (0, 300), \\[2mm] -\dfrac{300 - \Omega}{300} - 1, & \Omega \notin (0, 300). \end{cases} \quad (18)$$

In formula (18), Ω is the value of the process ratio.

The knowledge of the flow ratio table is shown in Table 4.

When the constraint characteristic value of the gating system is a discrete value in a non-numeric interval (i.e., qualitative description), the pressure loss characteristic of the gate can be divided into three cases: large (1), medium (0), and small (−1). The correlation function can be expressed as

$$(\Omega) = \begin{cases} 1, & \Omega = 1, \\ 0, & \Omega = 0, \\ -1, & \Omega = -1. \end{cases} \quad (19)$$

The matter-element is $\mathfrak{B}_{\mathfrak{U}}$ ($\mathfrak{S} = 1, 2, \ldots, \mathbb{Q}$) of the gating system. The correlation function value $(\mathfrak{B}_{\mathfrak{U}})$ of the evaluation condition $\mathscr{L}_{\mathfrak{U}}$ is finally applied with the goodness formula as

$$\&(\mathfrak{B}_{\mathfrak{U}}) = \mathscr{T}(\mathfrak{B}_{\mathfrak{U}}). \quad (20)$$

By calculating the goodness of each primary gating system scheme, the gating system scheme with the largest goodness value can be obtained.

Table 4: Current ratio table knowledge.

Plastic name	Flow ratio	Plastic name	Flow ratio
PP	280	PA	200–360
PE	100–140	POM	110–210
PS	280–300	PC	90–130

3. Experimental Results and Analysis

Flow analysis refers to predicting and displaying the filling state of the melt and the changes in pressure and temperature during the filling process by simulating the flow of the melt in the cavity. Under the filling condition of ensuring product quality, the operation of different design schemes under a series of relevant process conditions is simulated to help engineers predict possible defects before the formal processing of the mold, making it convenient to identify design shortcomings and improve them to determine the appropriate gating system, which reduces the risk and cost of mold opening. Flow analysis mainly includes filling time, pressure during speed/pressure switching, etc., which are not listed here. The filling time analysis results are shown in Figure 6(a). The pressure at the time of speed/pressure switching is referred to as V/P switching pressure. The main purpose of this analysis is to scientifically predict and compare the injection pressure required by all design schemes. The analysis results are shown in Figure 6(b).

Figure 6(a) shows that the three schemes are all filled well. The filling time of scheme one is 0.78 s. The filling time of scheme two is 0.78 s, and the filling time of scheme three is 0.44 s. Due to the large number of gates and large material flow in the third scheme, the filling can be completed in only 56% of the time of the first and second schemes, which is the optimal scheme for filling. Figure 6(b) shows the maximum injection pressure required for filling: Scheme one is 78.5 MPa. Scheme two is 70.1 MPa, which is 89% of scheme one, indicating that under the same process conditions, the injection pressure required by the second scheme is the smallest, and the loss to the injection molding machine is the least and low.

The warping deformation of plastic parts is mainly caused by the following three reasons: (1) Uneven volume shrinkage of plastic parts is mainly caused by the improper setting of holding pressure and holding pressure curve; (2) Uneven cooling is mainly caused by the improper design of the mold cooling system; (3) Molecular orientation is mainly caused by the improper design of raw material properties and the gating system. The purpose of this paper is to compare the degree of warpage deformation of the products under the same injection molding process conditions under the three design schemes to verify whether the design structure is reasonable. The final product warpage analysis results under all effects are shown in Figure 7(a). After separating the causes of warpage and evaluating the statistics, the results are shown in Figure 7(b).

Figure 7(a) shows that under the same injection molding process conditions, the maximum warpage deformation of the single-point pouring product is 1.23 mm. The maximum warpage deformation of the two-point pouring product is 0.85 mm, and the maximum warpage deformation of the three-point pouring product is 1.17 mm. It can be seen from the results and analysis and comparison that: (1) The two-point pouring design among the three schemes is relatively reasonable; (2) The positions where the products warp is exactly the same in all schemes, and thus it can be tried to separate the causes of warpage to further explore the causes of different warpage values; (3) Under the default process, the minimum warpage deformation solution is very close to the product process requirements, and thus you can try to optimize the injection molding process parameters of this solution to meet the product process requirements.

Figure 7(b) shows that uneven cooling is not the main cause of warpage deformation, and uneven shrinkage and orientation effects are the main causes of warpage deformation of plastic parts, of which the orientation effect is particularly important. Of all the solutions, two-point pouring had the best effect on improving fiber orientation. To sum up, the two-point pouring design scheme used above has good rationality and sufficient room for improvement, which verifies the feasibility and rationality of this design.

The parameters of the injection molding process have a huge impact on the molding quality of the plastic parts and play an extremely critical role in the entire injection molding production process. The injection molding process parameters mainly include molding time, injection pressure, clamping force, pressure holding curve, cooling time, cooling medium temperature, and mold temperature. Since all process parameters are closely related, it is difficult and time-consuming to achieve process optimization in such complex parameters. This injection molding process optimization is based on the principle of improving mold production efficiency, and the process optimization research is carried out on the premise of ensuring product quality.

The molding time mainly refers to the time it takes for the melt to fill the cavity. Whether this parameter is suitable has a huge impact on the production of plastic parts. If the molding time is too short, the required injection pressure will be large, which will increase the loss of the injection molding machine. If the molding time is too long, it is easy to cause short shots or insufficient filling at the thin-walled features of the product. At the same time, the production cycle of the product is prolonged, which makes the production cost increase invisibly. Therefore, the appropriate molding time is very important in the molding process of plastic parts.

In Moldflow molding time optimization, the gating system is not meshed when optimizing molding time due to the different shear heat algorithms used in the runner and the mold cavity. To explore the optimal molding time, this article uses Moldflow's "probe solution space" method. In this paper, the initial setting of the software is changed. The mold temperature and melt temperature are set as conditions, and the injection time is set as a variable. The optimal molding time of the plastic part is explored in the calculated function curve. After calculation, the preferred molding window obtained is about 0.23 s~0.43 s. After querying the function curve data, the optimal molding time of plastic parts is 0.32 s.
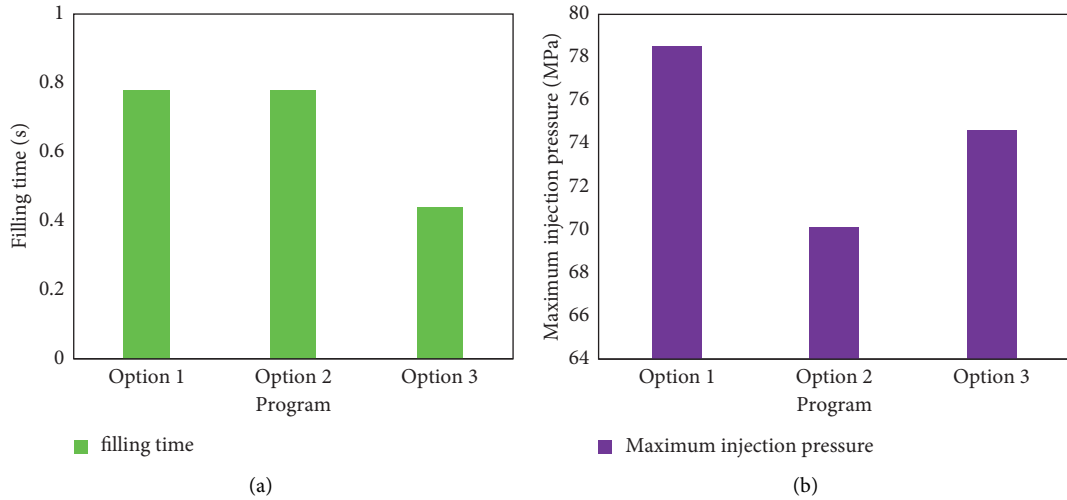
(a)

(b)

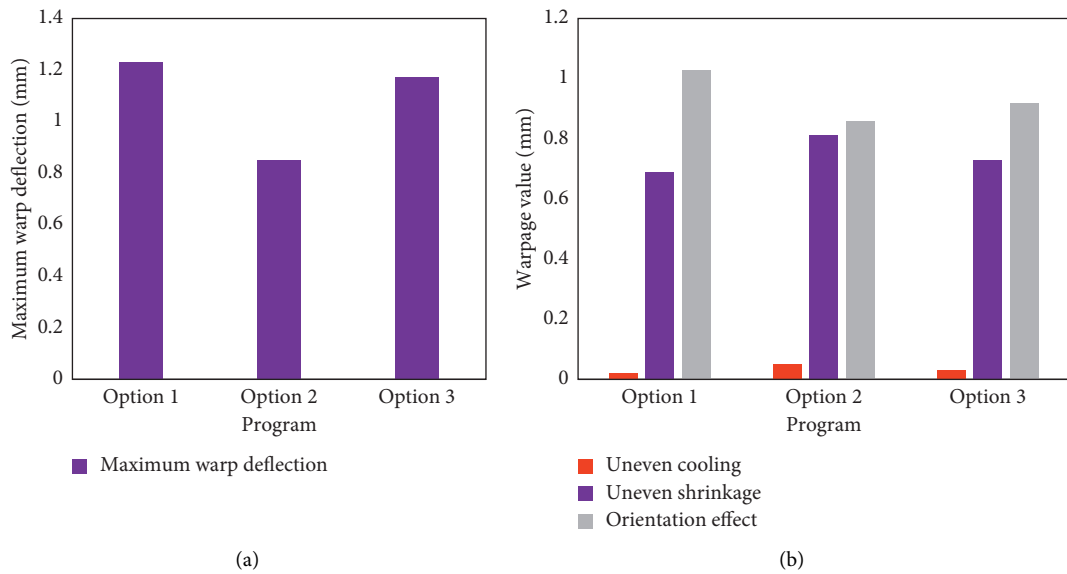Figure 6: Filling time and V/P switching pressure analysis.

(a)

(b)

Figure 7: Warpage deformation statistics of each scheme.

In order to verify the feasibility of this conclusion, this optimization will explore whether the three key aspects of minimum flow front temperature (as shown in Figure 8), maximum shear rate, and maximum shear stress (as shown in Figure 9) under the optimal molding time setting are feasible within the "preferred" molding time range.

Figure 8 shows that under the condition of using the optimal molding time of 0.32 s obtained above, the minimum flow front temperature of the melt is 290°C, which is very close to the recommended process for raw materials. When the molding time is higher or lower than 0.32 s, the minimum flow front temperature will be far away from the recommended process conditions for raw materials.

Figure 9 shows that the maximum shear rate and maximum shear stress corresponding to the optimum molding time of 0.32 s are well within the recommended process window for the raw material. After research,

calculation, and verification, the molding time parameter of the plastic part is finally optimized to 0.32 s.

The holding pressure curve is a function curve of the relationship between the holding pressure and the holding time of the injection mold. Holding pressure means that after the mold is filled, in order to prevent the molten material from returning to the mold cavity and make the plastic parts more compact to reduce the shrinkage and deformation of the product, the mold still needs a certain pressure to push part of the molten material into the mold cavity. This pressure will continue until the gate melt temperature falls below the transition temperature, a period of time known as the dwell phase.

The holding pressure curve is divided into two types: constant pressure holding pressure and segmented pressure holding pressure. Constant pressure holding pressure means that only one holding pressure value is used in the
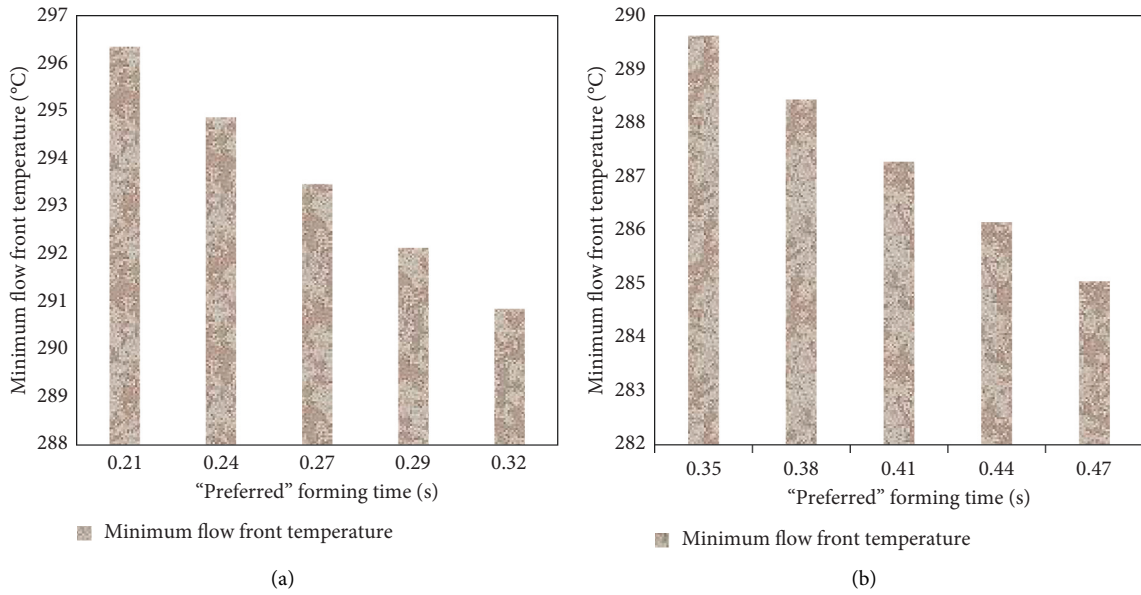
(a)

(b)

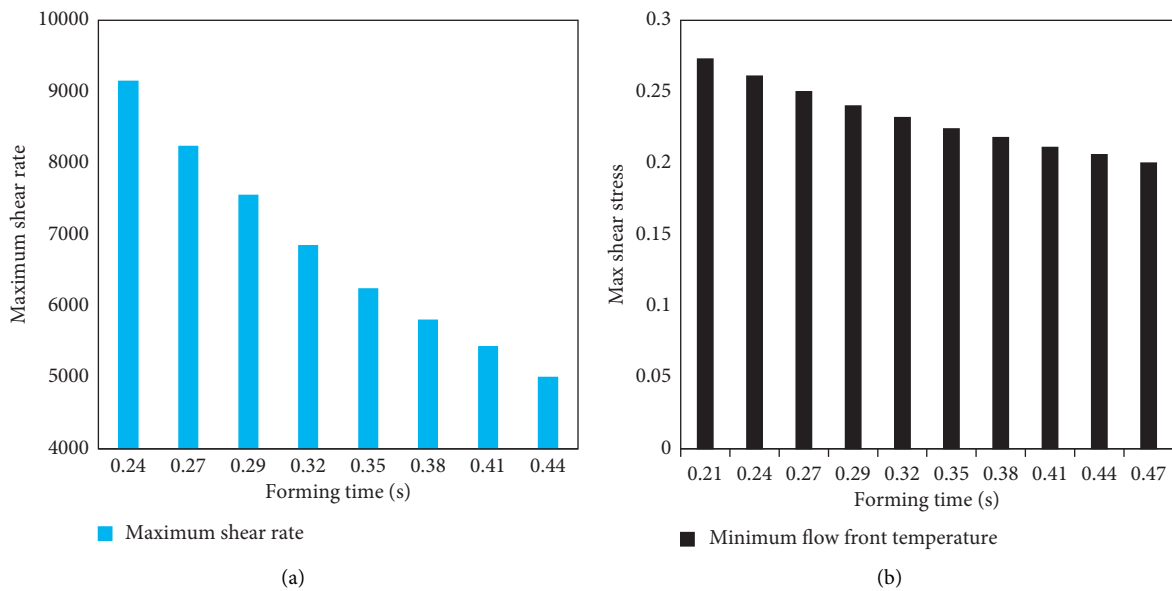Figure 8: Minimum flow front temperature.

(a)

(b)

Figure 9: Maximum shear rate and maximum shear stress.

entire holding pressure process, and the pressure directly decays to zero at the end of the holding pressure time. Segmented pressure holding refers to using multiple constant pressure holding pressure values in one pressure holding process, which is a curve that changes continuously with time.

The pressure data at the filling end of the plastic part are extracted in Moldflow. The maximum time of the pressure at this node is 0.9 s, and the time when the pressure decays to zero is 2.5 s. It is preliminarily defined as the constant pressure time, i.e., the starting time of the holding pressure decay, which is from the half of the sum of the maximum and minimum time of the filling end pressure minus the V/P switching time. The holding pressure adopts the default

value of Moldflow, and a preliminary holding pressure optimization curve is established as shown in Figure 10(a). After continuous adjustment and optimization of the holding pressure parameters, the optimal process parameters for this injection molding process are finally determined, and the holding pressure curve is drawn, as shown in Figure 10(b).

Figure 10(a) shows that during the initial optimization, the holding pressure is initially stable at 80%, and then gradually decreases to zero. Figure 10(b) shows that the difference between the packing pressure at the final optimization and the initial optimization is that there is an inflection point in the decrease, such that the packing pressure does not decrease at a constant rate.
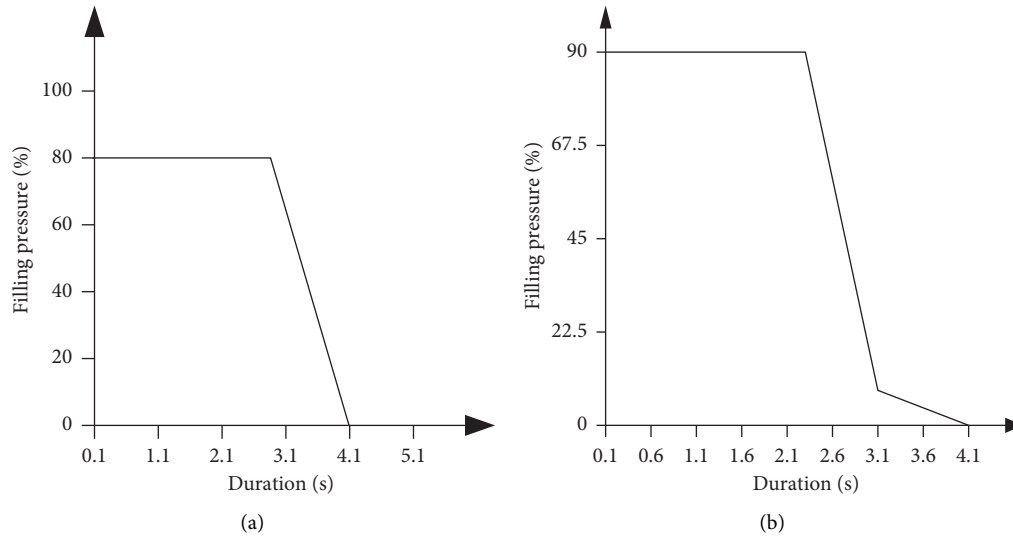
Figure 10: Packing optimization curve.

Table 5: Injection molding process parameter table.

Melting temperature (°C)	280	Holding pressure (MPa)	63
Mould temperature (°C)	75	Cooling medium	Water
Filling time (s)	0.46	Cooling temperature (°C)	50
Injection pressure (MPa)	70	Cooling time (s)	15
Holding time (s)	4.1	Mould opening time (s)	5

Through the optimization of the injection molding process parameters proposed above, the final injection molding process parameters are determined, as shown in Table 5.

The optimal injection molding process parameters in Table 5 are input in Moldflow, and finally the warpage deformation data of the product under all influencing factors can be obtained.

4. Conclusions

In this topic, a 3D digital model of the product is built in reverse using reverse engineering technology for the existing product, and an injection mold is designed for the obtained model according to the product process requirements. The design of the mold has been accurately researched and calculated, and a unique internal cutting and demolding mechanism is designed to achieve no casting marks on the surface of the product. At the same time, it ensures that the mold has stable working performance and long working life. The computer simulation of the injection molding process is deeply explored, and the related methods of process parameter optimization are proposed. The optimal process parameters of the mold injection process are finally obtained. The reliability of this series of parameters is verified, which can greatly reduce the workload of mold trial, and the results are reasonable and feasible. There is still a lot of research space for the optimization method of injection molding process parameters in the subject, which needs to be further discussed. In this paper, although temperature balance can be achieved in most parts of the mold, there is still a high temperature phenomenon in the thick part of a very small number of products. Therefore, in the next research, beryllium copper inserts can be used in order to improve the heat dissipation rate and achieve the temperature balance of the entire mold.

Data Availability

The data used to support the findings of this study are available from the corresponding author upon request.

Conflicts of Interest

The authors declare no conflicts of interest.

References

[1] Z. M. Huang, H. M. Kim, J. R. Youn, and Y. S. Song, "Injection molding of carbon fiber composite automotive wheel," *Fibers and Polymers*, vol. 20, no. 12, pp. 2665–2671, 2019.

[2] B. H. Seo, S. H. Noh, K. B. Choi, K. Jin, and S. U. Park, "A study on microcell injection molding process for automotive armrests," *Transactions of the Korean Society of Mechanical Engineers A*, vol. 42, no. 6, pp. 551–556, 2018.

[3] W. Guo, B. He, H. Mao, M. Zhang, L. Hua, and Z. Meng, "Mechanism of bubble formation in a combined in-mold decoration and microcellular foaming injection molding process," *Fibers and Polymers*, vol. 20, no. 7, pp. 1526–1537, 2019.

[4] H. S. Park, X. P. Dang, and X. P. Song, "Improving the cooling efficiency for the molding of a complex automotive plastic part

by 3D printing technology," *Transactions of The Korean Society of Automotive Engineers*, vol. 25, no. 4, pp. 508–515, 2017.

[5] Y. H. Cho, J. H. Kim, M. G. Lee, and B. M. Kim, "Prevent weld line defects on an automotive crash pad by local heating and cooling with 3D printed insert core," *Journal of Mechanical Science and Technology*, vol. 33, no. 5, pp. 2353–2361, 2019.

[6] G. Guo, "Investigation on surface roughness of injection molded polypropylene parts with 3D optical metrology," *International Journal on Interactive Design and Manufacturing*, vol. 16, no. 1, pp. 17–23, 2021.

[7] V. Contreras, F. J. Maturana, J. Poveda, K. C. Nunez, J. C. Merino, and J. M. Pastor, "Optimization of injection parameters to obtain selected properties on foamed PP with hollow glass microspheres and thermally expandable microspheres using Taguchi method," *Journal of Cellular Plastics*, vol. 57, no. 3, pp. 313–327, 2021.

[8] G. Wang, H. Yang, and Z. Lei, "Research on temperature and pressure responses in the rapid mold heating and cooling method based on annular cooling channels and electric heating," *International Journal of Heat and Mass Transfer*, vol. 116, no. JAN, pp. 1192–1203, 2018.

[9] C. L. Xiao and H. X. Huang, "Optimal design of heating system in rapid thermal cycling blow mold by a two-step method based on sequential quadratic programming," *International Communications in Heat and Mass Transfer*, vol. 96, no. AUG, pp. 114–121, 2018.

[10] J. K. Kim and E. S. Jeon, "Optimization of injection molding process parameters to improve mechanical strength of LFT specimen," *International Journal of Applied Engineering Research*, vol. 12, no. 13, pp. 3671–3676, 2017.

[11] Y. Song and Y. Zhao, "Research on development trend of mechanical automation technology based on technical characteristics," *Revista de la Facultad de Ingenieria*, vol. 32, no. 11, pp. 815–822, 2017.

[12] Z. Sun, Q. Wang, and L. Chen, "Unmanned technology-based civil-military intelligent logistics system: from construction to integration," *Journal of Beijing Institute of Technology (Social Sciences Edition)*, vol. 31, no. 2, pp. 140–151, 2022.

[13] M. İnal, S. Sahin, and Y. Sahin, "Optimization of the young's modulus of low flow polypropylene talc/colemanite hybrid composite materials with artificial neural networks," *IFAC-PapersOnLine*, vol. 51, no. 30, pp. 277–281, 2018.

[14] A. Uysal, "Influence of cutting temperature when drilling carbon black reinforced polyamides," *Materials Testing*, vol. 60, no. 6, pp. 627–632, 2018.

[15] J. Sreedharan and J. Ak, "Optimization of plastics injection molding process using response surface methodology," *International Journal of Engineering & Technology*, vol. 10, no. 1, pp. 139–148, 2018.