Design of Cooling System for Narrow Area of Automotive Injection Mold Based on Computer Technology

Liangqiang Wei1,*, Chengyun Peng2
1School of intelligent manufacturing, Chongqing Creation and Vocational College, Chongqing, China, 402160
2School of Material Science and Engineering, Chongqing University of Technology, Chongqing, China, 400054

*E-mail: liangcaia@126.com

Abstract. In the plastic injection molding process, the design of the cooling system plays a key role in the molding quality, molding cycle and production efficiency of the product. In this paper, computer technology is used to perform numerical simulation analysis on the injection mold of the automobile hubcaps. The mold temperature, the ejection temperature time of the plastic part and the distribution of the warpage of the plastic part are used as performance indicators to obtain the deficiencies of the cooling system design. Practice has proved that the cooling system optimization method is reliable and effective, and has certain reference value for the design of injection mold structure.

Keywords: Computer Technology, Injection Mold, Moldflow, Cooling System, Optimal Design

1. Introduction
Before the twentieth century, most industrial materials were mainly metal materials, but with the development of polymer materials and engineering technology, plastic materials are found everywhere, and they are widely used. Due to the lightweight, low cost, good molding properties, easy mass production, and recyclability of plastic products, the materials have always been loved by consumers[1]. Automotive interior trimmings are mainly composed of plastic parts, and more than 70% of plastic parts are processed by injection molds. So to speak, the quality of injection molds determines the quality of automotive interior parts. The production principle of injection molds is that the molten plastic particles are injected into the steel cavity processed according to the shape of the product by high temperature and high pressure. After cooling, the cavity can be opened to obtain the product of the desired shape[2-3]. When the plastic particles are cooled, a large amount of heat is released. This heat shall be discharged out of the mold cavity in time, otherwise which will cause many product defects, such as product color difference, surface scorching, warpage and deformation, etc. In extreme cases, the mold will be seized or even be scrapped[4]. To solve this problem, the general mold is processed through processing horizontal or vertical water channels in the mold core and cavity. Under normal circumstances, the distance between the water channel and the molded product is controlled at
about 20 mm, cooling water is passed through the water channel, and the purpose of heat dissipation is achieved by the flow of water \[3\]. However, this heat dissipation method has certain limitations. It is only suitable for molds which has enough product modeling space to arrange the waterway. The actual cooling time in the actual production occupies more than two-thirds of the whole period, so the design of the cooling system is very critical. A suitable cooling system allows the plastic products to dissipate heat uniformly in each area, further reducing the temperature difference caused by the different shape and structure of the product, and more importantly, it can effectively reduce the warpage caused by uneven heat dissipation, as well as reducing the molding cycle and improving productivity. Therefore, how to obtain the best cooling system for injection molding molds has become the focus of product design \[6\].

This article uses computer technology to carry out numerical simulation and analysis on the injection mold of automobile hubcaps, and conducts numerical simulation flow molding analysis to obtain the effect of different cooling system configurations on product quality, seek the best injection molding mold cooling system, and provide designers with theoretical reference.

2. Create analysis model based on computer technology
With the continuous development and growth of computer technology, many countries have begun to apply a large number of computer technologies in the design and manufacturing of plastic molds. Moldflow is the most commonly used in the field of plastic molding flow analysis. Here, the three-dimensional model is imported into the software system, and Fusion is used again to partition the mesh of the product, and if necessary, the mesh is repaired to create a product mesh analysis model, and then the best gate location is obtained through the Gate Location of Moldflow. It is best to choose the gate in the center of the product, followed by the outer edge, preferably not in the buckle. Under normal circumstances, the nameplate or car logo is identified on outer surface of the hubcaps. Of course, the surface quality must be guaranteed to be high, with the trace of gate forbidden. On the contrary, if the inner surface center is used as a gate, it will not only make the mold design and manufacturing complicated, moreover, the cooling water channel is not easy to be set up in the core of the product, seriously influencing the product's cooling and shaping and warpage, increasing the difficulty of taking out the condensate of the pouring system and directly bringing about the increase of cost. In summary, the outer edge of the inner surface of the hubcaps is used as the pouring position. Because the hubcap is a short and thin part, a mold with multiple cavities can be used in the mold design. In order to ensure the molding accuracy, it was decided to adopt one mold with four-cavity and balanced runner layout. In order to prevent the pouring part from staying in the sprue bushing when mold unloading, a banana-shaped tapered circular point gate is used. The typical point gate structure of the inverted cone shape is easily separated from the product when the mold is opened, and then the ejection mechanism provided under the gate is used to complete demolding.

3. Optimized design of cooling system
There are no fixed patterns and rules for the design of the cooling system of the injection mold. The design of the cooling system mainly includes the cooling mode of the mold, the size and position of the cooling medium flow channel, and so on. In order to improve productivity and reduce costs, most plastic product manufacturers generally use water cooling as the cooling method for injection molds, and the common cooling medium is normal temperature water, but it is the key how to determine the size and position of the cooling water channel and provide good uniform heat dissipation effect to cooling system design. If the designed cooling channel is too small, or the cooling channel spacing is too small, the cooling channel's heat dissipation function will not be able to perform well, which may cause local defects in the product, but the diameter of the water pipe should not be too large. A water pipe with excessively large diameter will slow down the flow rate, reduce the Reynolds number, and reduce the heat transfer coefficient; if too many cooling water channels are designed, it will not only increase the cost of mold manufacturing, but also bring difficulties to the design and manufacture of other mold structures; If the cooling water channel is designed not properly distributed, it will result in
an uneven temperature distribution on the surface of the mold cavity. For such problems, some designers adopt the method of evenly symmetrical distribution of cooling channels to solve the problem, but such approach will inevitably add a lot of costs to mold manufacturing, and sometimes ignore the enhanced cooling problem of the product's own heat load concentrated area, resulting in getting half the result with twice the effort.

3.1. Initial scheme design

Based on the consideration of the shape of the plastic parts of the automobile hubcaps and the structure of the related molded parts, the cooling system is initially set as shown in Figure 4. It can be seen from Figure 4 that the cooling water channels open four straight-through water channels above and below the product forming cavity respectively, cooling the fixed model cavity and the dynamic model core, and connected with a cooling hose outside, with the diameter of the cooling water channel of 8mm. The distance between the cooling water channel and the molding surface of the product is 15 mm, the distance between the cooling channels is 40 mm, evenly distributed on the cross-section of the injection mold. Moldflow in the form of injection molding analysis includes Fill, Flow, Cool, Warp, Stress, etc. For the setting of injection molding process parameters, there are mold temperature, melt temperature, injection speed, cooling time, holding pressure and holding time. In order to obtain information on product molding quality, MPI / Fill-Flow-Cool-Warp mold flow analysis is performed here, and database is built using moldflow software with reference to the recommended values of empirical data provided by General Motors of the United States to select the injection molding processing parameter, such as the melt temperature of 260 °C, the mold temperature of 60 °C, the holding pressure of 70MPa, the cooling time of 20S, the cooling medium of water with the temperature of 20℃. The Reynolds number of the inlet cooling water is 10000, that is, turbulent flow. Based on the main factors affecting the cooling system, some important simulation results are listed for analysis.

Figure 5 shows the mold temperature distribution of the initial cooling scheme. It can be seen from Figure 5 that the temperature of the buckle part of the hubcaps cools faster with average temperature of less than 50℃, but the temperature inside the end surface of the hubcaps is higher, that is, the temperature of the concave end of the large end of fixed mold is too high, which is about 12℃ different from the average temperature of the mold, which also shows that the freezing time of this area is relatively long. In addition, the mold temperature distribution range is wide, from 40.81℃ to 64.10℃, the maximum mold temperature exceeds the set 60℃, higher than the target mold temperature, it requires a longer cooling time, which can be seen from this cooling system that the partial thick wall is cooled insufficiently.

The time from the injection of the barrel to the completion of the solidification of each product in the cavity is the time of the ejection temperature. Figure 6 shows the initial solution

The time distribution of the ejection temperature of the plastic parts is about 21s at most. It can be seen from the figure that only the step flange part of the end surface of the hubcaps needs to reach 20s before it can be ejected, and it can be ejected in most areas within 10s for demolding, so for the ejection temperature distribution, the initial design plan is still feasible.

In the part where the mold cannot effectively arrange the cooling water path, the lower process hole (diameter D1) that is approximately parallel to the product surface shall be processed, grease is applied to the surface of the cooling rod (grease can improve the thermal conductivity of the cooling rod and can be disassembled as a cooling rod Lubricant), and then install it in the process hole, a cooling hole (diameter D2) is processed in the lower part of the cooling rod installed in the process hole, the cooling hole connects with the mold cooling water path, the lower part of this cooling rod becomes the low temperature end of the heat dissipation system, the heat released from the injection product rod can be passed through the cooling rod as a medium, which is continuously absorbed by the cooling water and sent out. In order to achieve the best cooling effect, the relevant dimensions of the cooling rods are shown in equations (1) and (2)
\[ D_1 = D + 0.1 \text{mm (The force makes the cooling rod disassemble and not easy to fall off)} \]  
\[ D_2 = D \times 2 \text{(Ensure the speed and reliability of thermal site conduction)} \]

Since the product needs to be matched with the automobile hub, the smaller the amount of warpage of the buckle, the higher the quality of the product will be. Figure 7 shows the distribution of the warpage of the plastic parts in the initial plan. It is known from Figure 7 that the maximum amount of warpage deformation of the product is 0.2483 mm, which is distributed in the clamping foot of the buckle, and the minimum amount of warpage deformation is 0.0293 mm, which is distributed in the central part of the end surface of the hubcaps, and the change range of the deformation amount is about 0.22 mm. Although the amount of deformation is within the allowable range of product error, if the maximum amount of deformation can be further reduced, or the range of variation can be reduced, it is more conducive to improving the accuracy of the product.

3.2. Improvement scheme design

In view of the design and analysis of the initial scheme, the cooling system needs to be further improved. Because the initial cooling scheme has a poor cooling capacity in the fixed mold cavity, the large end recess of the fixed mold is not only a place with heavy heat load, but also the place with the largest product wall thickness Therefore, it is necessary to strengthen the cooling and change of the straight-through cooling water to a part of the sinking cooling water channel. As shown in Figure 8, the cooling water in Figure 8 is closer to the end surface of the axle cover when passing through the fixed model cavity to solve the problem of the thick wall of the cooling system. Problem of insufficient cooling. In addition, in order to form the buckle of the wheel cover and facilitate the molding of the product, the molding structure of the moving mold needs to design a side extraction or inclined top mechanism. The initial cooling scheme directly passes the cooling water channel under the product, although it is conducive to the cooling of the product and the water channel.

The improvement program does not change the injection molding process parameter settings, and the physical parameters of the cooling channel are not changed. The distance between the cooling channel and the molding surface of the product is still 15 mm, the diameter is changed to 10 mm, the outside fixed mold is connected with a cooling hose, and the movable mold cooling water channel is set up directly on the moving formwork, MPI / Fill-Flow-Cool-Warp mold flow analysis is conducted again, and the analysis results are shown in Figure 1.

![Image](image-url)

**Figure 1.** Time of Improvement scheme Plastic part ejection temperature time

After the improved scheme, the mold temperature range is 34.91 °C ~ 49.76 °C, the maximum temperature is reduced from 64.10 °C to 49.76 °C, a decreasing amplitude of about 15 °C. The
temperature of the end surface area of the hubcaps with the heavier heat load is greatly reduced, which is significantly 60 °C lower than the set target mold temperature, shortening the cooling and setting time. It can also be seen from the figure that the minimum temperature of the mold is 34.91 °C, which is slightly higher than the original scheme, and the cooling speed of the buckle part is slightly lower than the original scheme, which is the result of the cooling water channel moving outside the mold core, but in general, the mold temperature distribution range becomes smaller, which is helpful to reduce the warpage caused by uneven heating and cooling. Therefore, the improved solution has a better cooling effect on the product.

After the improved scheme, the ejection temperature time of the plastic parts is about 3s earlier than the original scheme. The product cooling time span is reduced from 18.99s to 15.3s, bettering the cooling uniformity. It can be seen from the figure that only the step flange part of the end surface of the hubcaps needs to reach 18s to reach the ejection temperature time, and most regions can be ejected and demolded in less than 10s, similar to the original scheme.

Figure 2 shows the distribution of warpage deformation of the plastic parts after the improvement scheme. The maximum deformation amount is reduced from 0.2483mm to 0.2196mm in the original scheme, and the warpage deformation amount is reduced by 0.0278mm. In particular, there is a 11.5% change in the buckle position, and the amount of warpage deformation has decreased significantly. Because the improved cooling water channel is close to the place with heavy heat load and deviates from the area with small heat load, the configuration of the cooling water channel is elastically adjusted according to different heat load conditions, reducing the difference in the temperature distribution of the mold surface, and the cooling uniformity of product is also better than the original scheme, so the warpage caused by different mold temperatures and uneven cooling is improved.

![Image of warpage deformation](image)

**Figure 2.** Modification scheme Distribution of warpage of plastic parts

4. Summary
This article takes the plastic parts of automobile hubcaps as an example, uses computer technology to build a numerical analysis model, focuses on comparative analysis of the cooling system, and features the temperature distribution of the mold, the ejection temperature time of the plastic part, and the warpage of the plastic part during injection molding as the indicators, to find the shortcomings of the cooling system design, and adjust the cooling channel configuration elastically according to different thermal load conditions, processing characteristics of the forming structure, uniformity of the cavity surface temperature, and ejection time, and filter out appropriate cooling channel design. Finally, the
improved and optimized cooling system is used for injection molding production, and the method of using computer technology to optimize the cooling system is reliable and effective. It has certain reference value for the design of injection mold structure, and has certain economic and social benefits.

**Acknowledgments**
This work was financially supported by 2017 science and technology research project of Chongqing Municipal Education Commission: Research on key technology of design and manufacturing of conformal cooling plastic mold based on 3D printing (Project No.: kj1752487) and 2019 science and technology research project of Chongqing Municipal Education Commission: Flow analysis and process optimization of injection molding for automobile parts (Project No.: KJQN201905402) fund.

**References**

[1] Tang Y, Gao Z, Zhao Y F. Design of Conformal Porous Structures for the Cooling System of an Injection Mold Fabricated by Additive Manufacturing Process[J]. Journal of Mechanical Design, 2019, 141(10):1-8.

[2] Berger G R, Zorn D, Friesenbichler W, et al. Efficient cooling of hot spots in injection molding. A biomimetic cooling channel versus a heat-conductive mold material and a heat-conductive plastics[J]. Polymer Engineering & Science, 2019, 59(2):180-188.

[3] Clemente M R, Oliveira Pano M R. Introducing flow architecture in the design and optimization of mold inserts cooling systems[J]. International Journal of Thermal Sciences, 2018, 127(3):288-293.

[4] Paul C, Krug P, H. H. Müller, et al. Comparison of the horizontal diameter to a modeled area of traction in eyes with vitreomacular traction: is the diameter close enough to the truth?[J]. Albrecht von Graeffes Archiv für Ophthalmologie, 2018, 256(10):102-110.

[5] Fahruddin A, Muslimin M, Susilo G D, et al. Experimental study effect of diameter and number of spray nozzle holes on counterflow cooling tower to improve cooling effectiveness[J]. Matec Web of Conferences, 2018, 197(4):11-18.

[6] Paul C, Bertelmann T, Schumann R G. Letter of response to "Comment re: Comparison of the horizontal diameter to a modeled area of traction in eyes with vitreomacular traction: is the diameter close enough to the truth?"[J]. Graefes Archive for Clinical and Experimental Ophthalmology, 2018, 256(12):2485-2485.