

Original Research Article

Key mechanical structure design and finite element numerical simulation of side action mold

Tao Wang^{1,2,3}, Tao Zhang^{1,2,3}, Peng Lin^{1,2,3}¹Hebei Mechanical and Electrical Polytechnic, Xingtai, Hebei, 054000, China²Hebei Technology Innovation Centre for New Energy Vehicle Integrated Die Casting, Xingtai, Hebei, 054000, China³Xingtai Technology Innovation Centre for Mold Material, Xingtai, Hebei, 054000, China

Abstract: Taking a certain electrical appliance casing as the research object, a set of injection mold incorporating lateral parting and core-pulling mechanism was designed, which achieved collaborative CAD/CAE design. Based on the UG MoldWizard module, a fully parameterized 3D modeling was completed, including the parting surface, mold frame, pouring system, demolding mechanism, and cooling system. Based on the Moldflow platform, a comparison was made between direct and circular waterway. The results indicated that both schemes could meet the basic forming requirements. The circular waterway had slightly better cooling efficiency, volume shrinkage rate, and warpage deformation control, while the mechanical manufacturing cost of the straight waterway was lower. After comprehensive evaluation, the circular waterway scheme had more advantages.

Keywords: side action mechanism; finite element numerical simulation; cooling system; warpage deformation

1. Introduction

The plastic injection molding process is one of the main methods for electrical casing, and its mold design and manufacturing level directly determine the quality, production efficiency, and cost of the product^[1]. The geometric shape of the workpiece often includes special structures such as side holes, side indentations, threads, and inner protrusions. These features cannot be directly demolded through a single mold opening and closing motion direction during injection molding, and must be achieved through lateral parting and core pulling mechanisms^[2].

CAD/CAE collaboration is an efficient and reliable design method^[3]. Taking a specific model of electrical appliance casing as the research object, a full 3D mold structure design was conducted using the MoldWizard module of UG software, with a focus on explaining the motion principles and structural implementation of side core extraction and lifter mechanisms. Based on Moldflow software, finite element numerical simulation of the molding process was performed, with a particular comparison of the effects of two different cooling water circuit designs on mold temperature field, plastic part cooling time, volumetric shrinkage rate, and final warpage deformation.

2. Design of key mechanical structures for mold

2.1. Analysis of molding processability

The component was an electrical casing, as shown in **Figure 1**. The material was engineering plastic (ABS). ABS combines the chemical corrosion resistance and heat resistance of acrylonitrile, the high elasticity and impact resistance of butadiene, and the thermoplastic processing performance of styrene. It has excellent comprehensive mechanical properties and high surface glossiness, making it an ideal choice for manufacturing casings for household appliances and office equipment^[4]. The material has a density of 1.05g/mm³ and a shrinkage rate of 0.6%. Its melt flow rate is moderate and suitable for forming thin-walled complex products. According to UG software calculation, the volume of the plastic part was 11.64 cm³.

From the perspective of structural characteristics, there were multiple molding difficulties in this component.



Figure 1. Electrical casing.

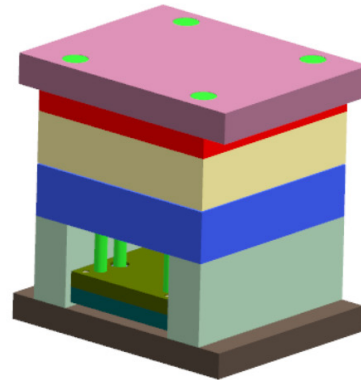


Figure 2. Mold base.

There were four rectangular through holes symmetrically distributed at the top of the workpiece, with a size of $20\text{mm} \times 5\text{mm}$. During design, it was necessary to ensure sufficient spacing between the forming protrusions to avoid small gaps in subsequent processing, which might cause the tool to be unable to process. The mold protrusions could be reasonably distributed on both sides of the core and cavity to increase the distance between small protrusions and facilitate mold manufacturing.

There was one lateral opening on the side of the workpiece, and its demolding direction was perpendicular to the main mold opening direction of the mold. This structure could be formed using a lateral parting core pulling mechanism.

There was a concave feature on the inner side of the workpiece, which hindered the direct detachment of the product from the core. This structure could use the lifter mechanism to simultaneously complete lateral movement during the ejection process.

2.2. Selection of mold parting surface

The selection of parting surface is the primary link in mold design, which directly affects the quality of plastic parts, the complexity of mold structure, and the difficulty of processing. Considering that this product had multiple through holes at the top and certain requirements for appearance quality, the main parting surface was designed at the maximum contour of the bottom edge of the component. This choice avoided leaving parting marks on the outer surface of the product, ensuring the appearance quality of the product.

2.3. Selection of mold base

The mold base is the skeleton of the mold, used to install and fix all formed parts, guide parts, ejection mechanisms, and cooling systems. Choosing the correct standard mold base can significantly shorten the mold design and manufacturing cycle. Taking into account factors such as cavity depth, ejection stroke, and cooling water circuit, this design selected a fine water nozzle three plate mold base, as shown in **Figure 2**. This mold base was suitable for sprue feeding and achieved automatic separation of solidified material and plastic parts in the pouring system.

The size of the fixed plate was $200\text{mm} \times 200\text{mm}$, with a thickness of 50mm. This thickness ensured that the mold cavity had sufficient strength to withstand the high pressure during injection, and provided space for the layout of the sprue channel.

The size of the moving plate was $200\text{mm} \times 200\text{mm}$, with a thickness of 50mm. Its thickness ensured the stability of the core and provided sufficient guidance and installation space for the movement of the lifter and ejector pin.

The thickness of the padding block was taken as 70mm. This thickness was reserved for a safety margin of 5-10mm during the ejection process.

2.4. Design of pouring system

The pouring system is the channel through which plastic melt flows from the injection molding machine nozzle to the mold cavity, and its design directly affects the filling behavior, pressure transmission,

and plastic part quality of the melt. The appearance of the electrical appliance casing required high standards and no obvious sprue marks were allowed.

This design adopted a three plate mold point gate structure, as shown in **Figure 3**. The gate was set at the center of the top of the workpiece. The diameter of the sprue was 1mm, and it could be automatically pulled apart during mold opening without affecting the appearance. By adopting a three plate mold structure, the automatic separation of the parts from the pouring system could be achieved, which realized fully automated production.

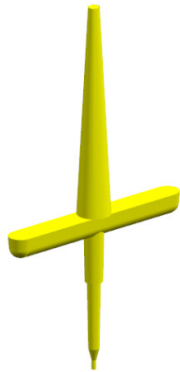


Figure 3. Pouring system.

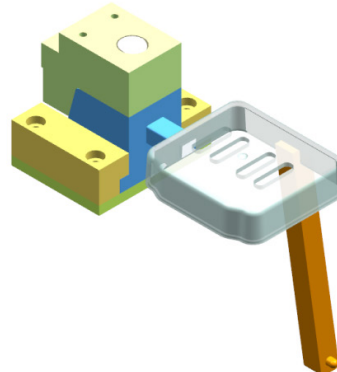


Figure 4. Side action mechanism.

2.5. Design of demoulding mechanism

This component included both outer side holes and inner side buckles, so the demolding mechanism was the core difficulty of the design. This mold required the use of both side slider and lifter mechanisms simultaneously.

For the side holes on the outer side of the component, a side core pulling with slider angle pin was designed, as shown in **Figure 4**. This mechanism was reliable in motion, which was a classic solution for solving the formation of outer holes.

A lifter mechanism was designed for the reverse buckle on the inner side of the component, as shown in figure 4. It was both a molded part and an ejector part. During the ejection process, as the ejection stroke increased, the inclined ejector rod pushed out the product upwards. Its top moved laterally to detach from the inverted area of the product.

To achieve smooth ejection, four cylindrical ejector pins were evenly arranged around the product.

2.6. Design of cooling system

The design of the cooling system is crucial for shortening the molding cycle and ensuring product quality. Uneven cooling can lead to high residual stress and severe warping deformation inside the plastic parts^[5]. However, the internal space of the mold was limited, and the layout of the cooling water circuit also needed to avoid many structures such as top rods and screws, making the design difficult. Based on this, two cooling schemes were proposed for preliminary structural design in this design.

Scheme 1 adopted a direct cooling water circuit with a diameter of 6mm. The significant advantages of this scheme were simple processing, low cost, and could be completed using a regular drilling machine.

Scheme 2 adopted a circular waterway with a diameter of 6mm. Theoretically, this scheme could achieve more uniform and efficient cooling, but it required the use of drilling and plugging processes, which was difficult and costly to manufacture.

The specific layouts of the two schemes would be accurately modeled and compared in Moldflow.

3. Finite element numerical simulation and analysis

3.1. Establishment of finite element model

Based on the Moldflow platform, STL format plastic parts and cooling water channel models were imported into the software. The grid type was Dual Domain. The global edge length was set to 1.79mm, and the final generated finite element model had 13142 elements. The overall setup met the requirements

for analysis accuracy. The models of the established straight through waterway (Scheme 1) and circular waterway (Scheme 2) were designed, as shown in **Figures 5 and 6**.

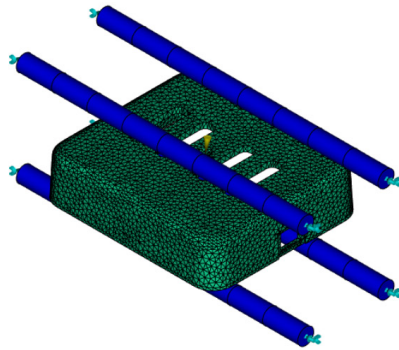


Figure 5. Direct waterway.

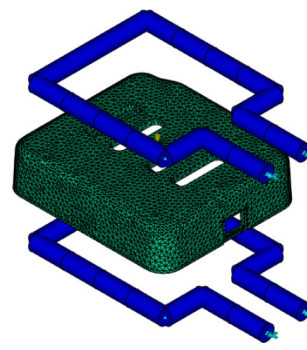


Figure 6. Circular waterway.

3.2. Determination of process parameters

ABS from CMOLD generic estimates company was selected. The molding temperature range of this material was 220-280 °C, and the mold temperature range was 25-80 °C. After comprehensive analysis, the injection molding process parameters was proposed as shown in **Table 1**.

Table 1. Process parameters.

injection temperature/°C	holding time/s	holding pressure/%	duration of cooling/s	water temperature/°C
230	30	80	30	25

3.3. Analysis of filling and flow

Based on the Moldflow platform, CAE analysis was completed using the above process parameters, as shown in **Table 2**.

Table 2. CAE analysis results.

scheme	time to reach the ejection temperature/s	time of filling/s	maximum volume shrinkage rate/%	maximum temperature of waterway/°C	warping deformation/mm
1	11.72	1.1	5.487	25.21	0.2724
2	12.22	1.098	5.360	25.42	0.2722

The filling analysis results showed that the melt flow was stable under both schemes. The melt filled the entire mold cavity in about 1.1 seconds. Due to the gate being located at the center of the top of the product, the melt diffused and filled outward in an approximately concentric circle, that ensured the balance of flow.

From the distribution cloud map of volume shrinkage rate, it could be seen that the maximum volume shrinkage rate of both schemes was between 5.36% and 5.487%. The volume shrinkage was mainly concentrated near the gate and in the area of wall thickness change. This was a normal phenomenon and could be compensated to a certain extent through the pressure holding stage.

3.4. Analysis of cooling

The time for scheme 1 and scheme 2 to reach the ejection temperature was respectively 11.72 seconds and 12.22 seconds. Direct water cooling was faster, but it resulted in uneven cooling. Although the total cooling time of the circular waterway was slightly longer, its cooling process was smoother and more uniform.

The maximum volume shrinkage rate of Scheme 2 was smaller. This indicated that the annular waterway could provide more uniform cooling, resulting in a more balanced pressure distribution and temperature gradient inside the mold cavity. This also reduced local excessive contraction.

The maximum waterway temperature for scheme 1 and scheme 2 was respectively 25.21 °C and 25.42 °C. This indicated that the circular waterway carried away more heat.

Warping deformation is the most critical criterion for evaluating product quality. The total warpage deformation of Scheme 1 was 0.2724mm, and scheme 2 was 0.2722mm, both of which were very close and at a relatively small level. The deformation of both schemes met the dimensional accuracy requirements of general electrical enclosures. The deformation value of scheme 1 was slightly larger by 0.0002mm, which

could be ignored. scheme 2 had a slight advantage numerically.

3.5. Analysis of air pockets

There was a risk of trapped gas at the filling end and in certain areas of wall thickness variation, which might lead to the formation of bubbles inside the product. Therefore, the exhaust groove must be set reasonably to guide the gas to be discharged smoothly.

4. Conclusion

A complex injection mold containing multiple core pulling mechanisms was successfully designed by integrating CAD and CAE technologies. Based on the UG MoldWizard module, three-dimensional parametric design from plastic part analysis to the entire mold has been completed. Based on the Moldflow platform, two cooling water circuit schemes were systematically simulated for flow, cooling, and warping. Comprehensive comparative analysis shows that scheme 2 (annular cooling water circuit) was determined as the final design scheme.

About the author

Tao Wang (1982-), male, from handan, Hebei, master's degree , professor, material processing CAD/CAE.

References

- [1] Cracogna R, Flamini M, Colladon F A, et al. Cost estimation of injection moulds for thermoplastic materials through a parametric approach[J]. The International Journal of Advanced Manufacturing Technology, 2026, 145 (5-6): 1-11.
- [2] Zhang Xuefen, Zhang Shaoyi. Design of injection mold for gas mask based on multi-directional side core-pulling mechanism[J]. Plastics Science and Technology, 2025, 53(5): 149-151.
- [3] Liu Yikao, Liu Yubo. Design of Injection Mould for Gauges Shell Based on CAD and CAE Collaborative Optimization[J]. Plastics, 2025, 54 (4): 162-167.
- [4] Zhang Lei. ABS modification and its application in appearance design of electronic and electrical products[J]. China Synthetic Resin and Plastics, 2024, 41 (5): 83-86.
- [5] Niu Jingli, Liu Jinqian, Ming Pingmei. Effect of mechanical cooling system on warpage deformation of ABS thin plate part[J]. China Synthetic Resin and Plastics, 2025, 42 (2): 56-59.