Design of Large Hot Runner Injection Mold for Interior Part Based on CAE Analysis

Ying Wang¹ᵃ, Weiting Huang¹ᵇ, Xuan Liu¹ᶜ
¹ Department of mechanical and electrical engineering, Foshan Polytechnic, Foshan, Guangdong, China
ᵃemail: wangying6191@163.com
ᵇ*Corresponding Author:email: outlook01@fspt.edu.cn
ᶜemail: 261753697@qq.com

Abstract: According to the shape and structural characteristics of an automotive interior plastic part, this paper has designed a large hot runner injection mold in accordance with the customers’ layout requirement, i.e., one mold and two cavities. It then analyzes the mold filling, cooling, and weld lines via the software, Moldflow, and hence determines a reasonable pouring system; a side core pulling mechanism, “lifter block + square push bar”, has been adopted for the large inverted part on the inner side of the plastic part, while a “lifter + insert” is organized for the two bayonet catches or inverted buckles on the inner side of the plastic part; the combination form of “ejector pin + lifter” has also been employed for the mold release system, with complicated mold structure. Practices prove that the mold structure is reasonable and reliable, and the production of plastic parts meet the quality requirements.

1. Introduction
Light weighting is becoming a top priority for the automotive industry, thus leading to the growingly obvious trend of substituting steel with plastic as the automotive parts. The production and quality of plastic products are closely related to the forming molds. The traditional mold designers used to rely on experience for mold design. But nowadays, with the injection molded products becoming increasingly complex, and the quality requirements is getting increasingly higher, and the dependence on experience alone often causes irreversible losses. Moldflow, a software with powerful simulation functions, allows mold designers to effectively analyze the gates’ number and positions, molding and mold defects such as weld and melt lines, sink mark indices, warping deformations and others. It also provides guarantee for the mold structure design, reduces the number of mold trials and repairs, shortens the production cycle, reduces costs and increases benefits[1-3].

2. Structural and Technological Features of the Plastic Parts
The inner plastic part in certain car model is made of nylon 6 adding 15% glass fiber, with a shrinkage rate of 0.45%, whose structure is shown in Figure 1. The structural characteristics of the plastic part are as follows: (1) its external dimension is 347 mm x 287 mm x 66 mm, its average wall thickness is 2 mm, and it is one mold with two cavities as required, which is a large mold and a hot runner is needed; (2) there are three inverted buckles on the inner sides of the plastic part, one mold with two cavities and a total of six buckles, which requires the use of “lifter + insert” organization, with a complicated mold structure; (3) the plastic part is an appearance part, molding defects are not allowed...
including sink marks, weld and meld lines and others\cite{4}.

3. Mold Design
The layout concerning mold cavities adopts one mold with two cavities, with the overall mold dimension of 900 mm × 700 mm × 531 mm, and the mold structure is shown in Figure 2.

3.1 Design for the Pouring System
When one mold with two cavities is employed and the mold size is large, with more reinforcing ribs on the inner surface of the plastic part, thin wall in some sections, and great resistance against the mold filling, it is difficult to fill the mold; the mold pouring system adopts a hot runner, according to the feature that the plastic part is relatively flat, while the gate adopts fan-shaped side gates; the grid divisions and the feeding scheme are shown in Figure 3, in which each plastic part adopts two side gates\cite{5}. The mold flow analysis presents that the gate scheme fills well and there is no molding defect, which is demonstrated in Figure 3.
3.2 Design for Molding Parts

When one mold with two cavities is employed and the mold size is large, the molding parts are designed as a whole insert, and the molding parts of the moving and fixed dies are designed as two cavities, which is convenient for processing. Since the material of the plastic parts is glass-fiber reinforced nylon, the friction of molten material to the mold cavity would be large in case of injection molding, the surface of the cavities in moving and fixed dies is nitriding heat treatment to improve the hardness, so as to improve its wear resistance. The core is installed in the square slot of the plate A and the die cavity is installed in the square slot of the plate B. Due to the large size of the cores and cavities, the cores and cavities would be positioned with wedge-shaped fixing block and fastened with hexagonal screws. In order to ensure the accurate alignment of the fixed and moving dies, the cores and cavities are designed to be interlocked with each other\textsuperscript{[6-7]}. The design for the cores and cavities is shown in Figure 4.
3.3 Design for Lateral Core-pulling Mechanism
The mold has six lateral core-pulling mechanisms, all in the inner side of the plastic parts, including two large inverted buckling parts which adopt a side core-pulling mechanism, that is, “lifter block + square push bar”. The maximum distance of the buckling is about 12 mm, and after calculations, the lifter needs to be designed for 10°. The remaining four inverted buckling parts are designed as “lifter + insert” side core-pulling mechanism, with the maximum distance of the small lifter of about 13 mm; the overall square lifter structure is used, and the calculated tilt should be designed for 12°. The design of lateral parting and core-pulling mechanism is shown in Figure 5.

3.4 Design for Ejection System
The mold has no flat surface, with several inverted buckles, so the combination form of “ejector pin + lifter” is used to release the mold. Each mold cavity has three inclined tops and eight ejector pins with diameter of 16 mm to eject the plastic parts. After the mold is broken, the injection molding machine pushes the ejector-pin plate which drives the push bar and the lifter to push out the plastic parts. After the ejection is completed, the injection molding machine first withdraws the ejecting force, the reset spring drives the ejecting system to pre-reset, and then the injection molding machine drives the part of the moving die to pair with the fixed die. When the reset lever touches the plate A, the reset lever acts to realize the precise reset of the mold[8]. The design of the ejection system is shown in Figure 6.

3.5 Design for Temperature Control System
The temperature control system of fixed dies adopts the combination form of “straight-through water
pipe + spacer water pipe”. The temperature control system of moving dies employs the form of “straight-through water pipe”. The straight-through water channel is convenient for processing and uniform cooling. The diameter of cooling water pipe is \$15\text{ mm}\$, and the diameter of the water well with spacers is \$25\text{ mm}\$. Large lifter, due to its size, also needs the design of cooling water, which adopts N type cooling form, with the cooling water aperture of 6 mm. The design of mold-temperature-control system is shown in Figure 7.

![Figure 7 Temperature Control System](image)

(a) Cooling system for moving dies  
(b) Cooling system for fixed dies

4. Mold Working Process

(1) Injection molding. The molten material enters the hot runner through the nozzle for injection molding machine, and then the mold cavity through the cold runner and the gate, before it fills the mold cavity, holds pressure, cools and solidifies.

(2) Opening the mold. When the molded products are cooled, the mold is opened and the injection molding machine drives the moving die through the mold base plate to open it, and the products are left on the core with the moving die during the process.

(3) Ejection. After the mold opening is completed, the injection molding machine pushes the ejector-pin plate which drives the push bar and the lifter to push out the plastic part to complete ejection.

(4) Resetting and closing the mold. After the ejection is completed, the injection molding machine first withdraws the ejection force, and the reset spring drives the ejecting system to pre-reset, and then the part of moving die is paired with the fixed die. When the reset lever touches the plate A, the reset lever acts to realize the precise reset of the mold. Then a cycle of injection molding restarts\[^{[9-10]}\].

5. Conclusion

A set of large hot runner injection mold in the layout of one mold and two cavities is designed for the structure and characteristics of an automotive interior plastic part. The Moldflow software is used to analyze the mold and determine a reasonable pouring system; this research aims to solve in advance the problems of weld lines, sink lines, filling and shrinking deformation that may occur in the injection molding; the inverted buckling part on the inner surface of the plastic part employs the lifter side core-pulling mechanism; the demolding system adopts the combination form of “ejector pin + lifter”, with complicated mold structure. Practices demonstrate that the mold structure is reasonable and reliable, and the production of plastic parts satisfy the quality requirements.

Acknowledgements:
This paper is supported by the Innovation Team Project of Guangdong Provincial Department of Education (2019GKCXTD005), Zhongshan Major Science and Technology Project (191007102629094), and Construction Project of Foshan Engineering Technology Research Center (2020001004036); 2018 Science and Technology Innovation Team of Foshan (Class B) “Application and Industrialization of Advanced Metal Matrix Composites” (2018IT100372); “Research Project on
Development Planning of Guangzhou Institute of Metallurgical Industry “(201800212).

References
[1] Xu Hui, Chen Hong, Hu Shaohui, et al. Design of injection mold for manifold cover based on CAE analysis [J]. Engineering Plastics Application, 2020, 48(9): 83–87.
[2] Hai Zhengping. Design of side-core pulling injection mold for auto-mobile air intake elbow [J]. Engineering Plastics Application, 2016, 44(2): 83–87.
[3] He Jingkui, Chen Hongtu. Design of large injection molds with hot runners for automobile side aprons [J]. China Plastics, 2008, 32(2): 134–140.
[4] Zhang Weihe. Design of injection molds for cars’ airtank-lefts [J]. China Plastics, 2014(4): 114–118.
[5] Li Zhengguang. Design of plastic cup slipcover and the injection mold structure [J]. Engineering Plastics Application, 2008, 36(5): 60–63.
[6] Yuan Xiaojian. Research on Flatness Error and Die Design of Bracket Plastic Parts [J]. Plastics Science and Technology, 2016, 44(12): 69-72.
[7] YAN Zhu-hui, LIU Bin. Design of injection mould for automobile lamp shell [J]. Die & Mould Industry, 2020, 46(12): 64-67.
[8] Cheng Guofei, Ding Ligang, Wei Wenqiang et al. Design of Injection Mold for U-shaped Base with Inclined Hole [J]. Modern Plastics Processing and Applications, 2020, 32(2): 46-49.
[9] XU Konglian, TAN Zanwu. Design of Hot Runner Mould with Two Complex Core Pulling Mechanisms for the Cover of Auto Filter Box [J]. Plastics, 2020, 49(5): 151-155.
[10] QIN X, SU X P. Design and Optimization of Injection Mold for PLC Shell Cover Based on Moldflow [J]. Plastics Science and Technology, 2018(9): 115-120.