# Design of Precision Injection Mould for Hot Runner of Automobile Back Door Inner Plate based on Front Die Core-Pulling

Huangying Wen

GuangDong University of Science & Technology, Dongguan 523000 China.

## Abstract

**Taking the inner panel of the Automobile Back Door as the research object, the design flow, design characteristics and key technology of the injection mold were analyzed, and a set of hot runner injection mold for the inner panel of the automobile back door was designed. Based on the analysis of die structure, forming parts, pouring system and side core pulling, the key and difficult solution of die was determined. The hydraulic cylinder +Slide Block core-pulling mechanism, inclined guide post +slide block and inclined push rod demoulding mechanism were used to solve the demoulding problem with many reverse buckles in different internal directions successfully, the position of Weld Line is adjusted successfully, the weld line is reduced, and the quality of plastic parts is guaranteed.**

## Keywords

**Inner Panel of Automobile Back Door; Injection Mold; Precision Mold; Core-Pulling Mechanism of Front Mold; Hot Runner of Sequence Valve.**

## 1. Introduction

The back door inner panel of the car is often used in the hatchback car or SUV model, which can seal the car and protect the safety of passengers.As shown in Figure 1, the inner panel of the back door of the car is located at the rear of the vehicle, which is injection molded by a mold and fixed on the back door of the car by means of screws and buckles.The inner panel of the car back door requires a certain degree of strength, safety, and beauty.This paper introduces in detail the key points and experience of the injection mold design for the back door inner plate of a brand automobile.

Fig.1 Position diagram of automobile back door inner panel

## 2. Plastic parts functional requirements and structural analysis

The maximum size of the inner panel of the car back door is 1106mm×613mm×185mm, and the average wall thickness is 2.5mm. It is known that the plastic part is a large thin-walled part. As shown in picture 2. Its characteristics and appearance requirements are as follows: ① Plastic parts for functional structure, general appearance requirements, parting surface is not allowed to have flash, dislocation clip wire,plastic parts are not allowed to have spots, weld marks, shrinkage and other defects; ② Plastic parts have 1 big bump perforation and 3 small bump perforations; ③ High surface finish and complicated internal structure; ④ There are 2 square hole undercuts on the side in the length direction, and 1 undercut at each end in the width direction, making it difficult to demold;as shown in picture 2.

Plastic parts material PP-T20, density 1.12g/cm3, shrinkage rate: 1.2%, it has been widely used in machinery, electrical, automobile, aircraft, ship and other manufacturing industries and chemicals[1].

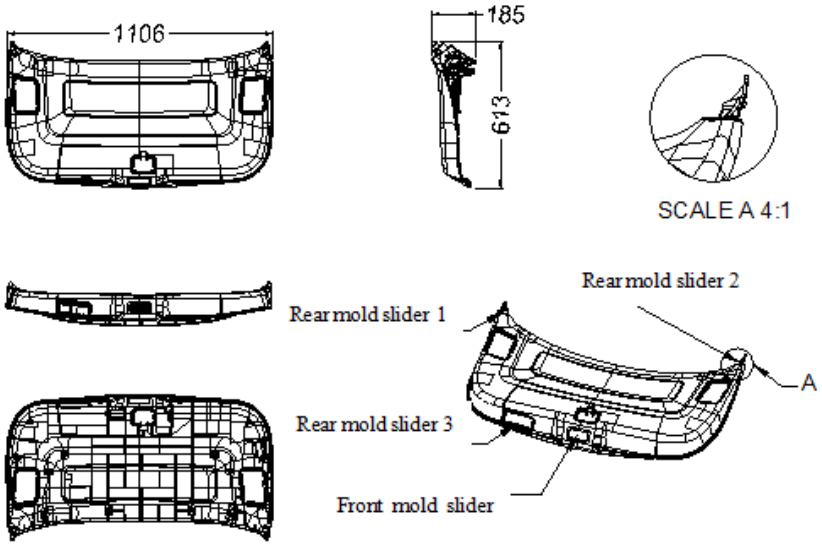



Figure 2 Plastic part structure

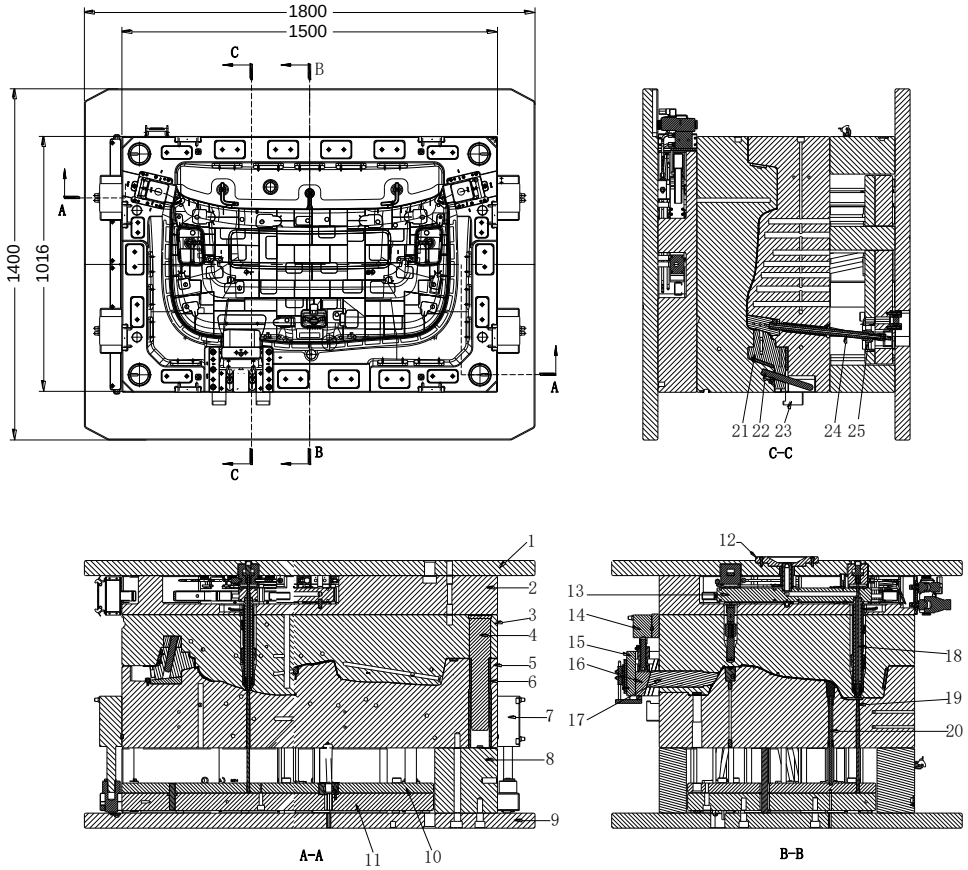



Figure 3 The structure of the injection mold for the inner panel of the back door

1.Front mold seat plate, 2.Runner pad, 3.Front template, 4.Guide post, 5.Post template, 6.Bushing, 7.Thimble surface cylinder, 8. Wurstite, 9.Rear mold seat plate, 10.Thimble panel, 11. Thimble bottom plate, 12.Positioning flange, 13. Hot runner plate, 14.Front mold cylinder, 15.Front mold slider base, 16.Front mold slider seat, 17.Front mold slider, 18.Hot nozzle, 19.Round putter, 20.Straight putter, 21.Rear mold slider, 22.Inclined guide post, 23.Slider baffle, 24.oblique push rod, 25.Inclined push rod holder.

## 3. Mold structure analysis

The plastic part is large in volume, thin in wall, inverted inside, and difficult to demold. Based on the above analysis, the mold adopts a hot runner injection mold structure, and the 6 needle valve hot nozzles are controlled by a sequence valve to enter the cavity through the cold runner and the side gate in turn. The mold structure of one mold and one cavity, the maximum size of the mold is: 1800mm×1400mm×1066mm, and the total weight of the mold is 20 tons. It is a large-scale plastic injection mold[2].The mold structure is shown in Figure 3.

### 3.1 Mold forming part design

Parting surface design as far as possible to ensure smooth, not easy to wear, no weak pointed steel structure; the insertion angle of the front mold and the rear mold is 8°, add a wear-resisting balance block to the insert surface and the bearing surface to prevent wear, ensure the accuracy of the clamping of the front and back molds, The parting surface other than the sealing surface is made 1mm to avoid the gap, which reduces the processing and assembly time of the mold assembly surface and improves the sealing performance of the parting surface[3].

On the premise of ensuring the strength and rigidity of mold parts, fully consider the feasibility of parts processing, and appropriately save costs.Both the rear mold insert and the front mold insert adopt an integral structure. The insert and template are integrally formed. The mold steel 718H is used for quenching and tempering treatment to prevent the integral structure from being too hard and cracking. The insert size is calculated to be 1500 mm×1020mm×330mm.

### 3.2 Gating system design

The plastic parts are not allowed to have spots, weld marks, shrinkage and other defects.When injection molding, the weld mark must be designed to the non-external surface or eliminated, which is one of the key and difficult points of this mold design. The traditional multi-point feeding at the same time, although the melt can fill the cavity, but due to the existence of weld marks, it is difficult to achieve the desired product quality[4].For this reason, the mold adopts a pouring system of "hot runner + cold runner + side gate feed", 6 sequential valve hot nozzles with feed, adjust the parameters during production to balance the melt filling of each gate,reduce the heat loss of the melt and reduce the weld marks to ensure the molding quality of the plastic parts[5].The location of the hot nozzle is shown in Figure 4.

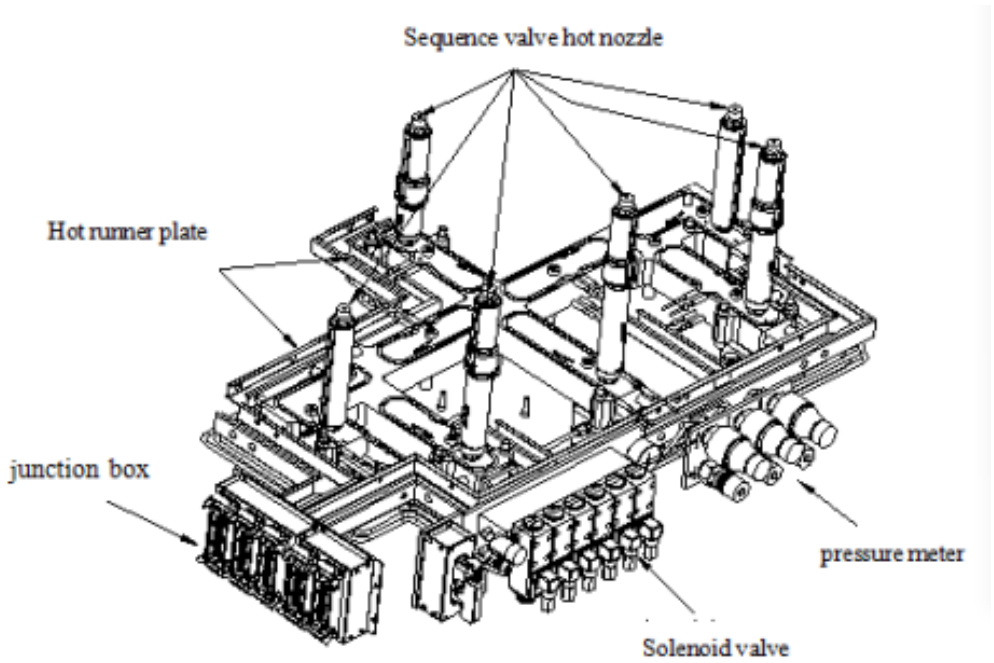



Figure 4 Mold nozzle

The mold parting surface is a 3D arc curved surface,it is necessary to design a platform at the feeding position as the feeding plane to assist the hot runner to the cold runner to feed, and it is conducive to processing, mold matching and production molding. As shown in Figure 5.

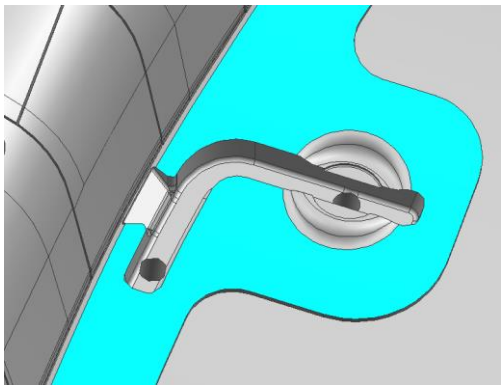


Figure 5 Feeding of the cold runner of the rear mold

### 3.3 Design of side core pulling mechanism

Analyzing the undercut structure of the plastic part, there are 4 parts that cannot be demolded directly, and there are also multiple undercuts inside.Based on comprehensive consideration, we decided to adopt: the front mold "hydraulic cylinder + slider" side core pulling mechanism, and the rear mold "inclined guide post + slider" "oblique push rod" side core pulling mechanism to break the undercut[6]. As shown in Figure 6 (a) (b) (c).

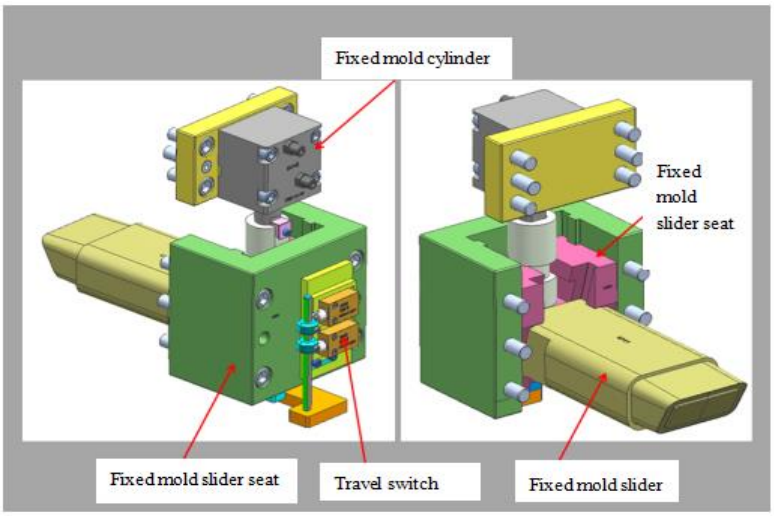



Figure 6 (a) Schematic diagram of the front mold slider core pulling

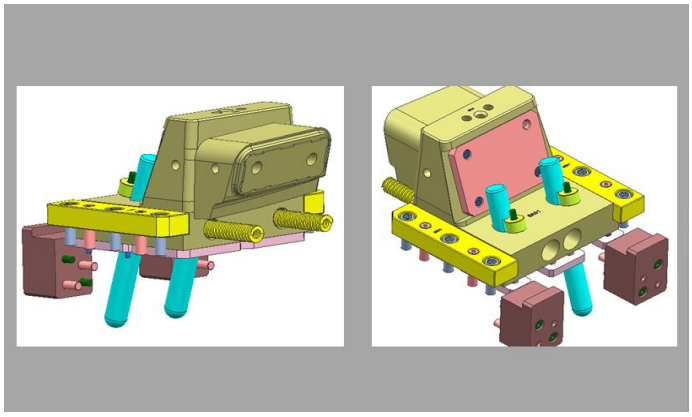


Figure 6 (b) Schematic diagram of the core pulling of the rear mold slider

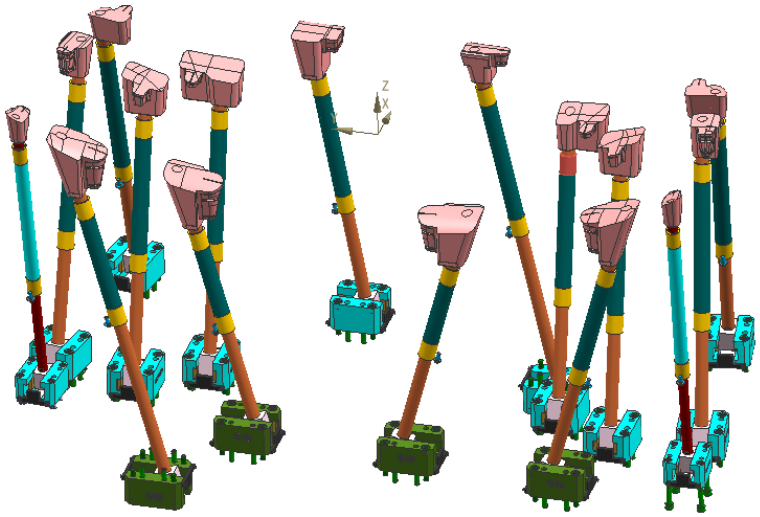


Figure 6 (c) Inclined push rod core pulling mechanism

### 3.4 Temperature control system design

The design of the temperature control system of the injection mold has a great influence on the production efficiency, and it is also related to the stability of the quality of the plastic parts.[7]In order to ensure stable mold temperature and uniform cooling of plastic parts, the cooling water circuit must be balanced and efficient.The front mold is designed with 13 groups of cooling water circuits, as shown in Figure 7 (a),most of the cooling water channels are combined with "straight through water pipe + spacer well",the bore diameter of the waterway is 10mm, the bore diameter of the well is 24mm, and the depth of the well is arranged at a distance of 25-35mm along the inner surface of the plastic part.The rear mold is designed with 11 groups of cooling water circuits, as shown in Figure 7(b), each group of cooling water circuit adopts the combination form of "straight water pipe + spacer well",the diameter of the waterway is 10mm, and the diameter of the well is 24mm.The depth of the well is arranged at a distance of 25-35mm from the inner surface of the plastic part.As shown in Figure 7(c), the slider is designed with 1 set of cooling water channels, which are 10-15mm away from the surface of the plastic part. Use the water collection block water pipe joint to connect the waterway, simplify the water pipe connection work.

Using the above temperature control system, the total cooling area has reached about 70% of the total area of the plastic parts.The temperature of the mold is balanced, the cooling is fast and sufficient, the injection cycle is shortened by about 10% compared with the same type, and the production efficiency is improved

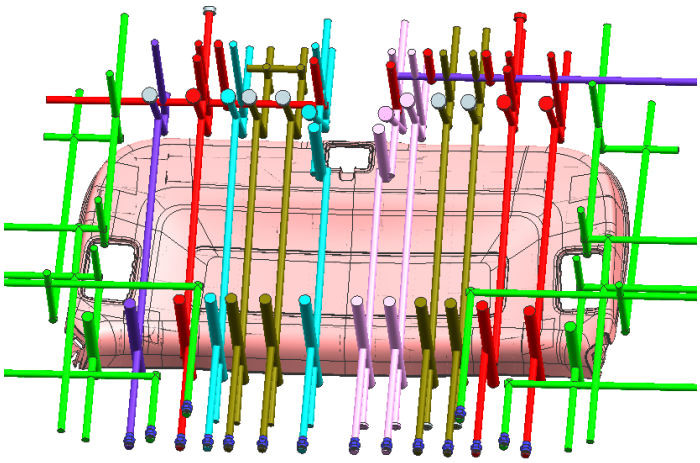


Figure 7 (a) Front mold cooling system

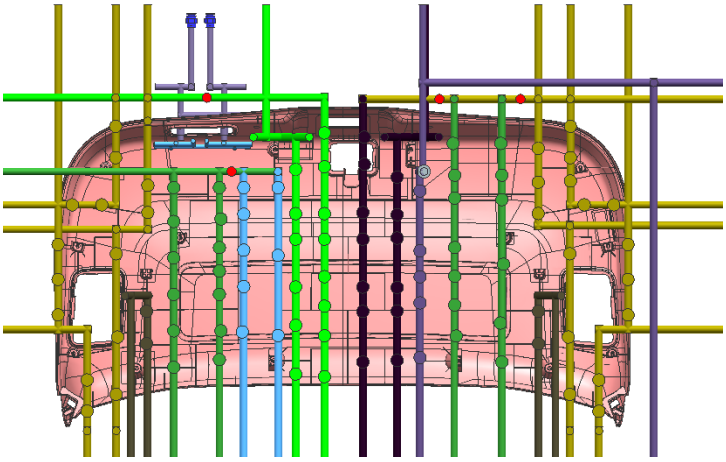


Figure 7 (b) Rear mold cooling system

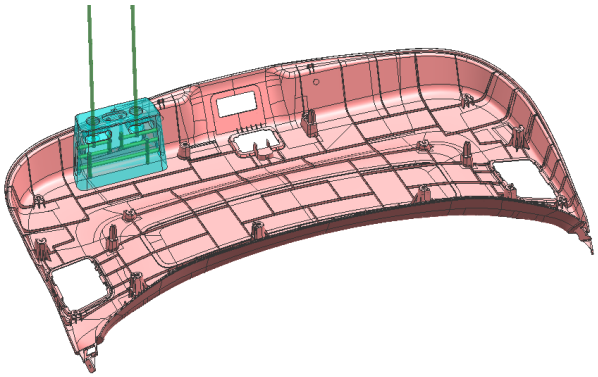


Figure 7 (c) Slider cooling system

### 3.5 Exhaust system design

The role of the exhaust system is in the process of injection mold cavity of the gas orderly and smoothly out of the cavity, so as to avoid plastic bubbles, poor filling, trapped gas burning and other defects, reduce the injection pressure, favorable plastic filling[8].The mold cavity has a large volume. In order to avoid the above molding defects, a good exhaust system must be designed to ensure the smooth flow of the plastic parts and the molding quality.The main part is the exhaust groove on the large parting surface, and the sliding parting surface, the oblique push rod and the straight push rod gap are supplemented, and the exhaust grooves are added at multiple corners. The width of the exhaust groove is designed to be 10-12mm and the depth is 0.03mm, as shown in Figure 8.

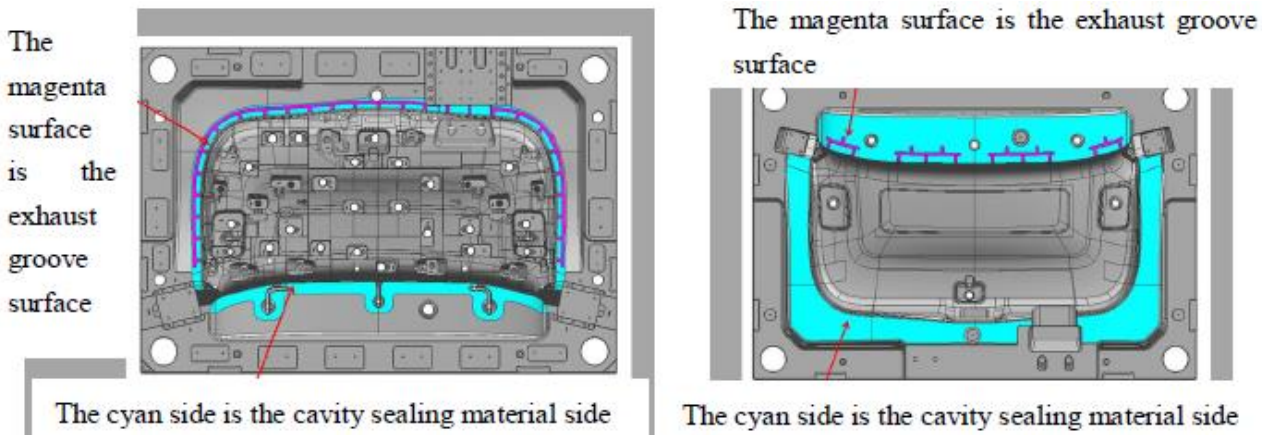



Figure 8 Exhaust slot

### 3.6 Design of demoulding mechanism

The plastic parts have a large tightening force. Larger straight push rods and oblique push rods are used when pushing out, and round push rods are used for special positions. The mould adopts "straight push rod +oblique push rod" combined release mechanism,because of its simple structure and convenient manufacture, the inclined push rod can not only realize the lateral core pulling, but also play the role of pushing out the plastic part[9]. After the mold is opened, the molded plastic part is pushed out by the inclined push rod and the straight push rod. The push rod fixing plate is mechanically pushed by the injection molding machine top rod through the K.O hole, and then reset under the action of the reset rod. As shown in Figure 9.

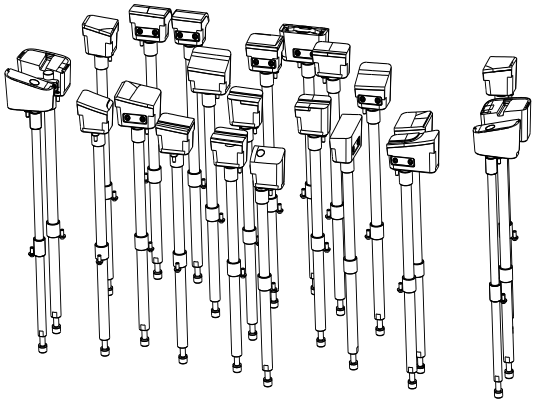


Figure 9 (a) Straight push rod

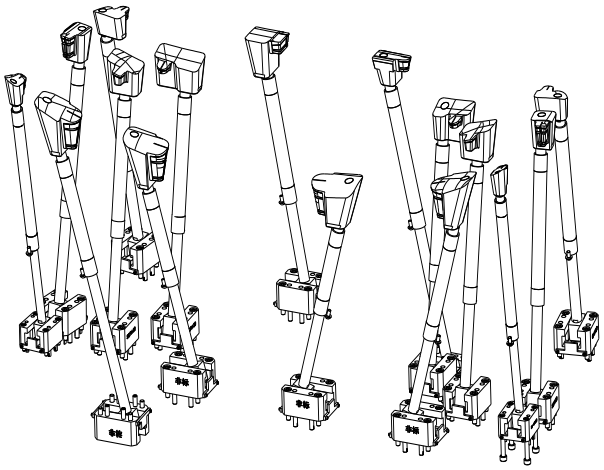


Figure 9 (b) Inclined push rod

### 3.7 Guiding positioning system design

In order to ensure the good guiding accuracy of the injection mold and a high mold life, 4 circular guide posts of Ø70mm×460mm and 4 circular guide sleeves of Ø90mm×Ø70mm×140mm are used to guide the mold opening and closing. 8° positioning slopes are designed around the periphery of the sealing surface of the front template 3 and the rear template 6 to ensure the tightness of the mold clamping before the moving template, as shown in Figure 10.

## 4. Mold work process

The mold working process is as follows:

(1) Injection molding: the plastic melt enters into the hot runner plate through the nozzle of the injection molding machine, then passes into the cold runner through the hot nozzle, enters into the mold cavity to fill and form, after pressure preserving, cooling and curing.

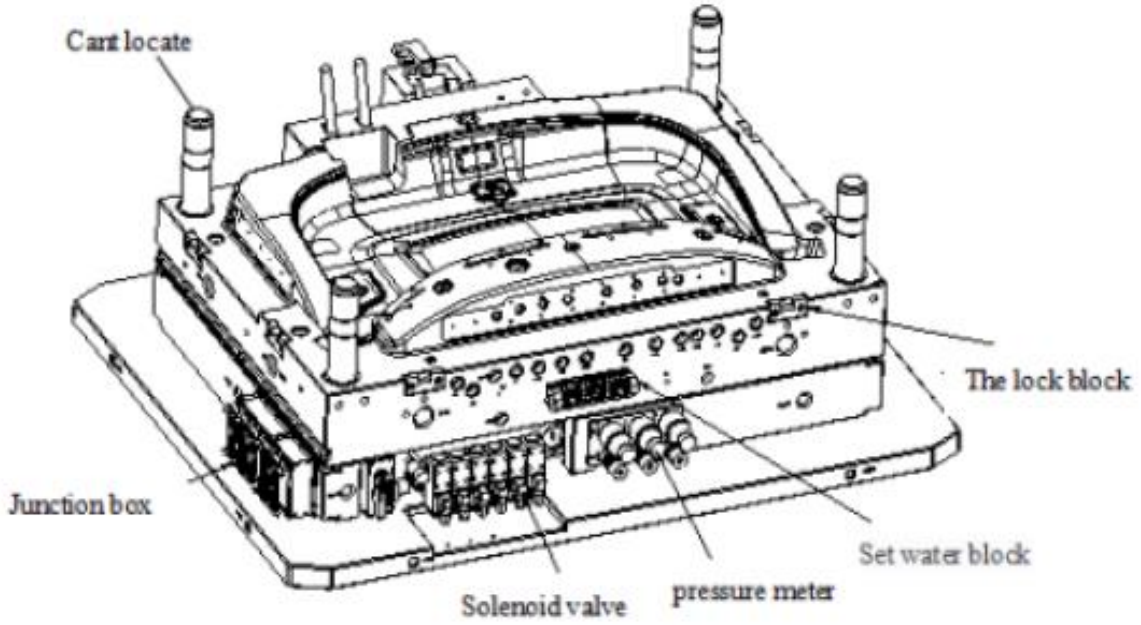
(2) Front mold core pulling: Before opening the mold, start the front mold cylinder 14, and the cylinder drives the slider 17 to move backward and release the reverse buckle.

(3) Mold opening: When the mold is opened, the injection molding machine drives the back mold to retreat to open the mold, and the back mold slide block is separated from the plastic part under the action of the inclined guide post.

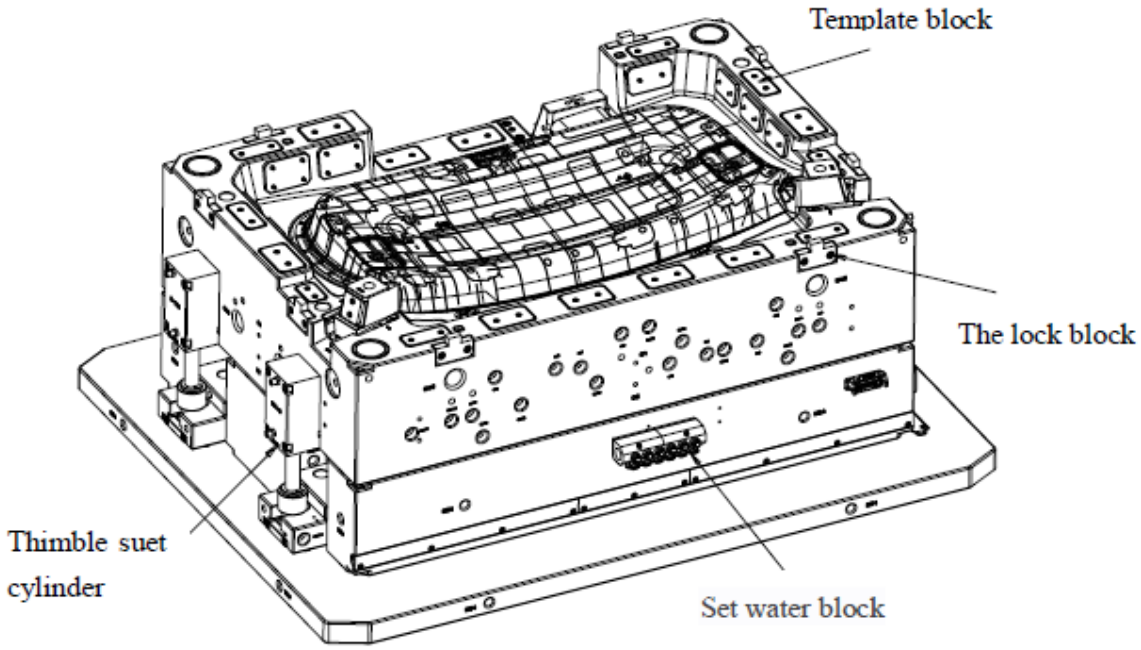
(4) Ejector: After mold opening, the ejector rod of the injection molding machine directly pushes the ejector bottom plate 11 and indirectly pushes the oblique and straight push rods to push out the molded plastic parts with a push distance of 140mm.

(5) Reset: After the plastic part is taken out, the ejector pin of the injection molding machine pulls the ejector bottom plate 8 to drive the inclined push rod and the straight push rod to reset.

(6) Mold closing: After the mold is closed, start the oil cylinder 14 to push the front mold slider 17 to reset, and wait for the next injection cycle.




(a) Three-dimensional view of the front mold




(b)Rear mold three-dimensional view

Figure 10 Three-dimensional view of mold

## 5. Conclusion

(1) The front mold adopts a reasonable insertion angle of 8° to solve the problem of accuracy of large mold clamping;

(2) Adopting the front mold "hydraulic cylinder + slider", the rear mold "inclined guide post + slider" and "oblique push rod" lateral core pulling mechanism, successfully solved the problem of difficult demoulding of internal irregular undercut;

(3) The sequential valve and hot runner are used for synergistic feeding to successfully adjust the position of the weld mark and reduce the weld mark, ensuring the molding quality of the plastic part;

(4) The uniform "straight-through water pipe + spacer type water well" combined temperature control system has successfully shortened the injection cycle by 10%.

The mold mechanisms are advanced and reasonable, running smoothly, safe and reliable after being put into production.

## References

[1] ZHANG Weihe.Injection mold design practical tutorial M. BEIJING: Chemical Industry Press, 2011:1-29.

[2] Qu Huachang.Plastic molding process and mold design M. BEIJING: China Machine Press, 1996:115-127.

[3] Deng Chenglin. A practical manual for automobile injection mold design. BEIJING: Chemical Industry Press, 2019:11-173.

[4] ZHANG Weihe.Design of large injection mold for left rear door panel of automobile J. China plastics, 2016,30(05):98-103.

[5] WAN Pengcheng.Design of injection mould gating system for automobile bumper and optimization of molding parameters M. Kunming University of Science and Technology Press 201158-75.

[6] Tian Shuzhu.Research on design of cooling control system of die and mould J. Die Making, 2018, 18(04): 52-54.

[7] Liu Fangfang, Zhang Weihe.Design of large-scale precision injection mould for rear view mirror base of automobile J. China plastics, 2020, 34(06):87-91.

[8] Huang Zhongyong,Wu Guangming, Huang Meirong. Design of injection mould for plastic shell J. Die Making, 2019,19(07):38-41.

[9] Zhang Bonan.Design of injection mould for mobile phone lower cover with fixed mould and Oblique Push Mechanism J. Equipment Manufacturing Technology, 2013,12:105-107.