Design of Injection Mould for Ventilation Net Cover

Kaiwu Cai

Huali College Guangdong University of Technology, Guangzhou 511325, China

171121932@qq.com

Abstract. In some parts of ventilation net cover as the research object, the injection mold design of the parts, the general steps of injection mold design, product analysis, material selection, design calculation, parameter analysis and calculation, the characteristics of mold process, overall design, mold design, mold assembly drawing, part drawing, mold check etc. The practical production has proved that the mold work is stable, the actual production requirements are reached, and the production is carried out. The produced plastic parts are of good quality, it can be used as a reference for the design personnel of injection mold.

1. Introduction

With the rapid development of computer technology and its application in plastic mold technology, plastic mold technology has also been developing rapidly, and has changed the traditional methods of mold design and manufacturing. At present, there are some differences in the actual production process and the production process of plastic mold on the theory, the process of development of our country at the present stage, plastic mold design and production has been seriously strong plastic products, with the development of the times in daily life and production equipment are widely used, so the strengthening of plastics the mold design level has a positive impact on the development of plastic products industry in China in the future. The design and analysis of the mould for the ventilation net hood are summarized, and the key points and advanced technology are summarized. By referring to the design data of the related mould [1], the injection mold design is carried out.

2. Analysis of material and process of injection molding products

Analysis of plastic types and process of using materials, ABS plastic universal thermoplastic molding plastic performance is good, good fluidity; molding shrinkage of small (usually 0.3~0.8); specific heat capacity of low density r=1.03~1.07g/cm^3; higher plasticizing rate in the barrel, in the mold solidification forming rapidly. The cycle is short, disadvantage is the larger absorbent, therefore, must be fully dry before molding, can be formed in the plunger or screw type horizontal injection machine. The ventilation net hood drawing as shown in Figure 1, the technique and structure analysis of plastic parts: plastic parts of the size, shape rules, parting surface selection on the surface of the plastic parts plastic parts, but there are many holes, so need to arrange four side core pulling mechanism, the gate spoke gate [2].

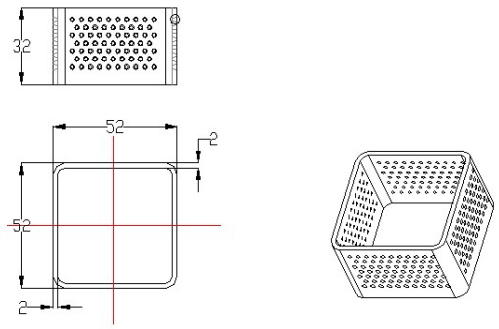

Figure 1. Engineering drawings for ventilation mesh parts.

3. Mould design and related calculation

3.1. Structural design of forming parts

Because concave and convex parts are directly contacted with high temperature and high pressure plastics, and they are repeatedly rubbed with plastic parts when demolding, it is required that the concave and convex parts have enough strength, stiffness, hardness, wear resistance, corrosion resistance and low enough surface roughness. The fixed mold structure, the mold to the core structure, so the dynamic model with local inlaying concave mould [3]. The dynamic model structure, the integral dynamic model material waste is too large and the cutting amount is large, the structure of the mold in the almost no such structure, mainly the whole embedded modular punch and punch. The overall embedded punch is chosen here.

3.2. Working dimensions of forming parts

The working dimension of the forming part is the size of the plastic part directly by the specified mould and the moving mould. The precision of the plastic parts is directly affected by the precision of the working dimension of the fixed and moving modes. Factors affecting the dimension of plastic shrinkage affected parts, impact wear punch and dimension of manufacturing tolerances of the punch and use process and other factors; calculating the mould size: usually, the punch and the work size according to the plastic shrinkage, determine the manufacturing tolerances of concave and convex mould parts and wear three factors. The main factors affecting the dimensional accuracy of plastic products in forming process include the following aspects: dimension accuracy of molding parts such as cavity and core when manufacturing. The core and cavity are integrated with the embedded structure. The error of manufacturing and assembly will affect the size accuracy of the product. Fluctuating value of plastic shrinkage rate: Generally speaking, for process conditions, injection pressure increased and shrinkage rate decreased. Temperature increased, shrinkage rate increased, holding time prolonged, shrinkage rate decreased, mold release speed increased and shrinkage increased. The influence of product structure on shrinkage is as follows: inserts are less than embedded parts, thin walls are thicker than walls, shape is more complex than shape, shrinkage is reduced to a certain extent, while the shrinkage of radial dimension is larger than that of thickness direction [4]. The average shrinkage rate of the ABS plastic is 1.07, with the average shrinkage rate of the mould's working size. The plastic part adopts the 6-stage precision, the manufacturing tolerance of the concave and convex mould to take the 1/3 - 1/6 of the tolerance of the plastic parts. The radial size of the large core of the moving model is 48.5mm, and the depth is 32.3mm.

4. Design of injection mold structure and molding parts

4.1. Design of the gating system

The design adopts the latent gate, and the section of the diversion channel is trapezoid. Mainstream Road Design and calculation, the main channel is divided into two sections, one section in the gate

sleeve, a section in the fixed plate insert. Based on past experience, the mainstream design is the cone, the cone angle is 4 degrees. Because under the premise of forming and stripping, the cone angle is small and the waste of the runner handle is less, which improves the yield of the raw material. The inner wall of the main channel should be smooth and the surface roughness of Ra is 0.40~0.20, and the direction of polishing is parallel to the direction of demoulding. The length of the main channel is shorter in the case of the thickness of the template, the better it is, generally less than 60mm. This design takes 47mm. The design and calculation of the positioning ring: according to the size of the positioning hole of the injection molding machine, the positioning ring with a diameter of 100mm is selected. The submarine gate, the cross-section shape and size similar to the gate of the gate, which in addition to the various characteristics with point gate, the feeding part of the general election in the side or the back of a workpiece shelter, does not affect the appearance of the product can be used at the same time, the two plate mold simple. The runner design, runner shape common round, hexagonal, trapezoidal, U shape, half round, rectangular and several kinds of hope, easy processing, and the same in the channel length and the flow volume under the condition of flow resistance and heat loss are the smallest cross-section shape. The design uses wheel auxiliary branch runner, because this flow path is only machined on a template, which saves mechanical processing costs, and heat loss and resistance loss are not large, so it is a commonly used form. In addition, the diversion channel should be as short as possible and the volume is small, because the loss of melt temperature and pressure flow through the flow passage is small. Due to the contact cooling of the injector nozzle and the cold mold, the cold front well design often causes a low temperature material at the front end of the nozzle. In order to get rid of the cold material, there is usually a cold material well across the main road, so that the cold material will not enter the shunting channel and the cavity. The cold material well of the mould used in the horizontal injection molding machine is located on the moving mould corresponding to the end of the main channel. At the bottom or around, it often makes zigzag hook or lateral grooves. It is the cold material well which can pull the main road condensate out of the main channel and stay on the moving mould when it is divided into the mold.

4.2. *Design of model oriented mechanism*

To ensure the accuracy of the shape and size of the cavity, the plastic mould should be closed according to a certain direction and position. Therefore, a guiding and positioning mechanism must be set up. The most common guiding positioning mechanism is to set 2 - 4 guiding columns and guiding holes [5] around the mold cavity. Therefore, 2 pairs are used in the design, and the guide column is located at the edge of the moving mould. The length of the guide column is L 85mm. At the same time, the length of the guide column is higher than the height of the mould face 6 - 8mm, and the front end of the guide column is made of a hemispherical shape, and the angle is needed, so that it can enter the guide hole smoothly. Take R=2mm; and the taper of the front end is Q=150.

The guide column should have a hard and wear-resistant surface, the core part of the tough and not easy to be broken, so the use of low carbon steel (steel 20) (0.5~0.8mm), carburizing after quenching, hardness more than 55HRC (HRC56~60).

The tolerance coordination is that the transition coordination H7/k6 is adopted between the installation section and the template. The guide section and the guide hole use dynamic coordination H7/f7. The surface of the fixed section is Ra0.8um, and the surface of the guide section is Ra0.4um Design of guide sleeve

In order to fit with the guide column, the inner diameter of the guide sleeve is d=25mm, d1=35mm, S=6mm, R=1mm.

The material is wearable material, and its hardness is 5 degrees lower than that of the guide column. It can be quenched with carbon tool steel with a hardness of HRC50-55. The coordination tolerance is H7/n6 with the outer surface and the template hole tight, and the surface roughness of the inner and outer surfaces can be Ra0.8um.

4.3. Design and calculation of demoulding mechanism

The calculation of demoulding force: the ratio of the thickness of the circular wall to the diameter of the inner hole is greater than 0.05 in the cross section.

$$F_{d2} = \frac{2\pi r \cdot E \cdot \varepsilon \cdot L \cdot \left(f - tg\varphi\right)}{\left(1 + \mu + K_2\right) \cdot K_1} + 10$$

The design of the launch agency: There are many pieces of plastic conical hole walls, forming four blocks using the slider, drives the slider four separate, pushing plate extrusion of plastic parts. According to the specific situation of this design, the slider is separated and the plastic part is introduced after the slider is separated.

4.4. Design and calculation of core pulling mechanism

(1) The core is S = S0 + (2~3) mm, and the side hole depth S0 = 2mm is S = 2 + 3 = 5mm, and the angle of the inclined hole is 18^0.When the parting core is finished, when the slider and the inclined pin leave each other, the slider must stay at the position of rigid separation, so as to smoothly enter the inclined hole of the slide block when clamping, so we must set up the slider positioning device. So use the wedge in the wedge.

(2) The design of the slant pin: because the slider is used to form a conical hole, the required pulling force is small, so the diameter of the slant pin is 14mm according to the actual situation. The length of the slant is L = 61 mm.

(3) The guide groove is designed and the guide groove is a member for supporting the slider to carry out the movement of the core. The matching between the guide groove and the slider is smooth, not too loose and too tight, so it is simple and convenient to use straight groove. The slippery surface should have enough hardness (HRC52~56) and should be slightly lower than the slider. In order to make the slider move without bias, the sliding surface of the slider should have enough length, and it is best to be 1~1.5 times the width of the slider. When the pull-out motion is stopped, the sliding surface does not always remain in the guide slot, but the length in the guide slot should not be less than the width of the slide block, so that it is difficult to reset the slider. The structure of the guide groove is T - shaped groove, and the sliding surface of the slider is 70mm.

(4) The design of the wedge block, the wedge block is to prevent the slider movement in force when the molding parts (usually can not rely on the oblique pin lock slider), it should be better than the wedge angle of oblique guide column angle 2 degrees to 3 degrees. Because the oblique angle of the slant pin (slider) is 180, the wedge angle of the cuneiform block is 200. The effect: when the mould is closed, the wedge angle contact block angle, due to the tight wedge angle ratio slider bevel value, the wedge block drives the slide speed faster, at this distance in the slide completely by the wedge block and push locking; when the mould is opened, the wedge block wedge leave the sliding block bevel angle, with oblique pin and inclined block holes have a certain pore, so the slider completely by the oblique pin smoothly pulling out. If the wedge angle of the tight block is the same as the slope angle of the slider, the error caused by the inclination may cause the failure to open the mould.

When the mould is closed, the slider is completely pressed by the wedge, so the precision of the combination of the inclined guide column and the slider hole does not need to be very high, and the F9 coordination precision is generally taken.

At the same time, the wedge block should have enough surface hardness (HRC52 to 56) to avoid scratching and deformation. The mold is to open rectangular holes on the template, and then insert into rectangular wedge blocks.

According to the concrete size of the slider and mould, the size of the wedge can be determined, and the width of the wedge is 60mm.

5. Analysis of the action process and structure of the mould

This mold is a single parting mold, and its action process is as follows: after completing a series of actions such as glue injection, pressure maintaining and cooling, the mold is finished in the mold cavity. When the mould is opened, the movable template is separated, and the product stays on the side of the movable template under the action of the tightening force. The four sliders on four sides are separated from the wedge block first and then open to the four sides under the effect of the slant pin, and the products are exposed and the core pulling action is completed. The sliding block is separated from the slanting pin after the end of the core drawing. The push plate and the push rod fixed plate move forward, the push rod pushes the product into the model cavity, completes the mould release, and then takes the plastic part and the runner coagulant by the manipulator. After the demoulding is completed, the push plate and the push rod fixed plate are returned to the original position from the effect of the reset rod, and the opening process is completed. In the mold closing, the slant pin closes the four sliders to the middle, and the slider is locked under the action of the wedge block to prevent the high pressure from acting on the slider and forcing the slider to move outside. When the slider is closed, the moving mould is closed and pressed under the action of the clamping force to carry out the injection molding of the product. At this point, a period of action is completed. The detailed assembly drawings of the mold are as shown in Figure 2.

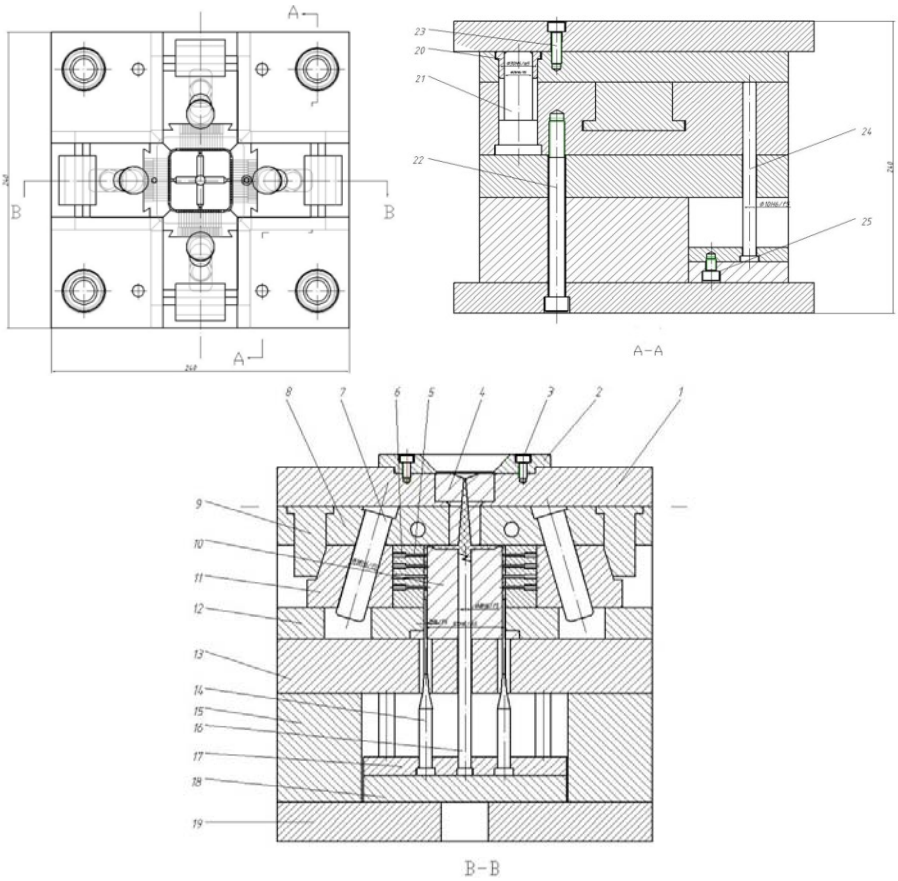

Figure 2. Mold assembly drawing.

1-fixed plate fixed plate; 2- positioning ring; 3- screw; 4- gate sleeve; 5- slider head; 6- small core; 7- inclined guide column8- fixed mold plate; 9- lock module; 10- large core; 11- slider tail; 12- moving template; 13- supporting plate; 14- push rod;15- supporting plate; 16- pulling rod; 17- push material fixed plate; 18- pushing plate; 19- dynamic mould fixed plate; 20- guide bushing; 21- guide pillar; 22- fixed bolt; 23- six angle screw; 24- reset bar; 25- screw

6. Summary

From the analysis of plastic parts to the determination and parameter checking of injection molding machine, the calculation of mold design, the structural design of mould, the checking of injection machine parameters, the determination of the thickness of cavity wall, the thickness of the bottom plate and the design of cooling system, the analysis and research are made. The design requirements of plastic parts, production volume, volume and quality of plastic parts were defined, and the parameters and selection of injection molding machine were considered in detail. The structure of the mold is subdivided, and the mould frame and gate sleeve are selected according to the requirements. For the large injection quantity, the clamping force, the mold and the injection machine installation part of the related dimensions are checked, the structure is stable, plastic to meet the design requirements.

Acknowledgments

This project is supported by Foundation for The Youth Innovative Talent Project of Guangdong general colleges and universities in 2017 (2017KQNCX254).

References

[1] Huachang Qu, Mengling Wu, Anna Shi. The history of plastic molding process and mold design third edition Qu Chang. Beijing: Higher Education Press, 2014:8.
[2] Beijing medi Technology Co., Ltd.. Pro/ENGINEER Chinese wildfire version 5 mold design course (value-added version). Beijing Machinery Industry Press, 2017:3.
[3] Mingchang Liang, injection molding technology and production management. Beijing: Chemical Industry Press 2014:1.
[4] Haitao Dong, the process flow of plastic mold design and manufacture [J]. Modern manufacturing technology and equipment. 2015 (04)
[5] Zhenyu Li, design of plastic cap injection mould for different kinds of core pulling mechanism [J]. Plastics technology. 2015 (11).