Design and Checking Analysis of Injection Mold for a Plastic Cup

Xuebing Li

Guidaojiaotong Polytechnic Institute, Shenyang 110023, China
251336431@qq.com

Abstract. A special injection mold was designed for the structural characteristics of a plastic cup part. The mold was simulated by Moldflow software and verified by calculating the stripping force, the pulling force and the clamping force of the mold so that to determine the appropriate injection parameters. It has been proved that the injection mold is effective and practical in the actual producing and can meet the quality requirements during the course of using it, which solved some problems for injection molding of this kind of parts and can provide some reference for the production of other products in the same industry.

1. Introduction

With the continuous development of various engineering plastics, injection molding is used more extensively in the manufacturing field to make various products for meeting requirements. In order to produce high quality injection products economically, we must consider the problem about injection mold comprehensively. The quality of injection mold directly affects the efficiency, quality and cost of molded products. It plays a very important role in the forming of injection products. Not only the surface quality, precision of plastic products, are completely determined by the mold, but also the internal quality and efficiency of forming products are also affected by the mold. So how to fast and standardize designing injection mold has become the primary problem to develop the technology.

Here the plastic cup in this article was made of polypropylene including antioxidants and UV Absorbents. Its shape and structure are shown in Figure 1.

2. Designing injection mold

The drawing of injection mold to this cup is shown in Figure 2. According to the actual situation of the plastic part, the parting surface is set as the location shown in Figure 3. In order to avoid the occurrence of sharp corners between plate and the slide block, the side should be rounding at the bottom of the parting surface. Due to repeated contact and collision between mainstream, high temperature plastic melt and nozzle of injection machine, the main channel is often designed as a detachable gate sleeve, which is made of steel T10A and hardened to 50 ~ 55HRC. Steel T10A is...
suitable for manufacturing various tools which has a high wear resistance, a certain toughness and a sharp edge cutting under poor conditions.

![Figure 1. part drawing of plastic cup](image1)

![Figure 2. Mold structure drawing](image2)

1. Water injection well choke 2. Stopper 3. Spring 4. Steel ball 5. Thin wall pipe 6. Right slider 7. Water injection well choke 8. Base plate of fixed mold 9. Sprue bush 10. Locating ring 11. Hexagon socket head cap screws 12. Wedge block 13. Inclined guide column 14. Left slider 15. Water injection well choke 16. Core 17. Sealing gasket 18. Water injection well choke 19. Plug screw 20. Interface to pressure pump

3. Calculation and analysis

(1) Calculating demoulding force

According to $\delta \cdot d^{-1} = 0.0278 < 0.05$, Here $\delta = 2.5\text{mm}$, $d = 90\text{mm}$; Therefore, the plastic part is a thin-walled plastic part and because the section of the part is circular ring, so
where S is average shrinkage rate of plastic (\%), E is modulus of elasticity of plastic (MPa), l is the tolerance length of the plastic to the core (mm), \( \phi \) is demoulding slope of mold core (°), \( \mu \) is Poisson's ratio of plastic, \( \delta_i \) is the average wall thickness of a plastic part (mm), \( f \) is the static friction between the plastic and the core, \( f = 0.1 \sim 0.2 \), A is the projection area of the mould core in the blind hole in the demoulding direction and \( k_2 = 1+f\sin\phi \cos\phi \approx 1 \).

The material of this plastic part is PP and the relevant parameters are \( S = 1.75\% \), \( E = 950\)MPa, \( l = 40mm \), \( \mu = 0.392 \), \( A = 276.46mm^2 \), \( \phi = 3 \) degree. It can be calculated from (1) that the demoulding force is approximately 81.65KN.

(2) **calculation of core pulling force**

The core pulling force can be obtained in Mould design and manufacture; that is,

\[ Q = lhp_2 (f_2 \cos\theta - \sin\theta) \] (2)

where \( l \) is the circumference of the section shape of a movable core packaged with a plastic part. \( h \) is the depth of the cavity part. \( p_2 \) is extrusion pressure per unit area that plastic parts to core (MPa), generally 8~12MPa. \( f_2 \) is the friction coefficient between plastic and steel generally 0.1~0.2. \( \theta \) is the draft angle of a side hole or side concave(°). It can be calculated from (2) that the core pulling force is approximately 125189.1N.

(3) **Checking of the clamping force for the mold**

When the melt high pressure plastic fills the cavity, it will generate a great thrust, which should be less than the nominal clamping force of the injection machine. Otherwise, it will generate overflow phenomenon. That is,

\[ F \geq PA \] (3)

where \( F \) is the nominal clamping force of the injection machine(N). \( P \) is the injected pressure into the cavity during injection. Here \( P = 25\)MPa. \( A \) is the sum of the vertical projection area both the plastic and the pouring system on the parting surface (mm²). Here \( A = nA_1 + A_2, n = 1 \). \( A_1 \) is Projection area of a single plastic part on a parting surface (mm²). \( A_2 \) is the projection area of a runner (including a gate) on a parting surface. It can be calculated from (3) that the thrust force in the mold is approximately 1506.9KN. So the clamping force of the injection molding machine can meet the requirements.

(4) **Analysis of plastic parts by Moldflow**
1) preprocessing

1) importing the model, analyzing the type, and select the material

Import the cup model, choose the mesh type as the surface model, double-click the fill bar on the left side of the task bar, select the analysis type as "gate location", then select "TaiwanPP" as the material and 6331 as brand, take the "immediate analysis". As shown in Figure 4., the blue display is the best gate location from the results of the analysis.

![Figure 4. optimal gate position](image1)

2) creating the gating system and cooling system

Place injection position at the center of the cup’s bottom. Choose gate location center as the mainstream position. Set the entrance diameter as 3mm, taper angle as 3 degrees, the length as 52mm. Create a cooling system by hand, place the inlet and outlet of the water and the inlet of the coolant, so as to finish creating the gating system and cooling system. As shown in Figure 5..

![Figure 5. cooling loop](image2)

2) Output analysis Results

Carry out analyzing the cup by "cooling + flowing + warping" in Moldflow. Found from the analysis, the largest clamping force of injection molding machine is 7 tons, the maximum injection pressure is 1.8MPa, the filling time is 1.5s. When the filling stage is 1.48s and the flowing rate is 33.46 cubic centimeters per second, switch speed and pressure. Holding pressure begins from 1.5s, at 11.49s pressure releases and at 28.14s pressure ends. The surface temperature of mold is set as 60 degrees Celsius. The ejection condition is set as 100 degrees Celsius and injection pressure is set as 120MPa.

It can be seen from the report: 1) The deformation caused by different cooling is different from each other, and lengthening the cooling water channel at the slide block, as shown in Figure 6., will make the deformation shrink. 2) When the filling time is 1.5s, rotate the plastic part and choose different points on parts. As shown in Figure 7., it’s ok for the changed time between the filling time is not more than 0.2s. 3) The position which generate the gas holes is on the parting surface or in the gap between the left and right slider, or the edge of the cup. Where it is easy to exhaust, so there is no problem. 4) Although it is inevitable to get rid of the weld marks. There are still some weld marks in the position of the handle. We can lessen the weld mark to make it unobvious by increasing the speed, temperature and pressure. 5) The desire to the clamping force is much less than the maximum clamping force that the injection molding machine owns, so it is feasible. 6) finding the temperature deviation on the wall of loop tubes is very low shows that the cooling effect is very good.

4. conclusions

The results show that the various calculations of theories and advantages in mature CAD and CAE software, taking the mold design, product design and experimental verification before manufacturing into a comprehensive consideration, can reduce the cost on material and time caused by
some unreasonable design, can increase the reliability of the product design. It can help avoid the
defects on designing, shorten the cycle for producing.

The method here for design and checking analysis to this injection mold can provide some reference
during the injection processing. But there are still some deficiencies in the study of some parameters
for this mold. So further research must be needed in some related aspects.

References

[1] CAO Jiang-dong, LU Li-xin. Optimum Design of Injection Mould of Hairdryer Shell Based on
Moldflow, J. PLASTICS, 2010, 39(1): 122-125.

[2] Liu Bin, Wu Xi. Analysis and Design Method of Conformal Cooling Channel of Injection Mould,
J. Engineering Plastics Application, 2017, 45(8): 123-127.

[3] SONG Ke, HU Qing-chun, JIANG Xiao-ping. Optimization Design of Injection Mold Cooling
Channels Arrangement, J. China Plastics Industry, 2014, 42(12): 45-48.

[4] ZHAO Zhenfeng, SHI Xianzhang, WANG Jing, SHEN Changyu, LIU Chuntai. Experimental
analysis of pipe network for injection mold, J. CIESC JOURNAL, 2009, 60(12): 3148-3151.

[5] LI Xi-ping, ZHAO Guo-qun, GUAN Yan-jin, LI Hui-ping. Analysis and optimal design of heating
and cooling process for rapid heating cycle injection mold, J. JOURNAL OF PLASTICITY
ENGINEERING, 2009, 16(1): 196-200.

[6] Jeppson R w. Analysis of Flow in Pipe Networks. Boston: Butterworth publishers, 1976: 1-86