The injection mold design of handle parts based on HS CAE

ZHANG Yan Qin[1, a], DENG Zhao Hu[2,b]

[1] Electromechanical Engineering Department

Dongguan Polytechnic

Dongguan, China

[2] Mechanical and Electrical Engineering Department

Guangdong Polytechnic College

Guangzhou, China

[a]zhangyanqin410@163.com , [b]cowboy-dhw@yahoo.com.cn

Keywords: CAE analysis; mold design; submarine gate

Abstract. In this paper, the process was discussed with CAE analysis and CAD design together to complete plastic mold design, with handle parts as example. CAE simulation can be used to analyze the best gate location, filling, pressure maintaining, cooling, warping and other problems. With the combination of mold CAD and CAE technology, it can detect the design defects early, improve part quality, reduce the tryout number, so as to improve productivity and reduce costs.

Introduction

The model of handle parts was shown in Fig.1. The dimensions of this part were 190×120×20mm, and the thickness was 2.5mm. After forming the plastic parts, the inside and outside surface should be smoothly, the pin imprinting should be uniform, the parting line should not exit flash, and no sink mark. The product yield was 20-30 million one year, using ABS as material.

Fig 1.Handle part

CAE analysis

CAE analysis of plastic molds including filling analysis of runner, gate and mold cavity, pressure maintaining analysis, contraction analysis, cooling analysis and warping analysis.

Filling analysis. The design of runner and gate system is the first element of the molding product quality. In this paper, a mold with two cavities with submarine gate was designed to complete CAE simulation analysis, because the outer surface of the part was not allowed to have traces of the gate.

The pressure which runner and gate need can befound out by analyzing the behavior of the molten plastic after injecting into the mold cavity. It can be also found out that if the pressure is too large or not and whether the design of the runner and gate size was reasonable. The part may have some defects such as short shot, gas porosity, welding line, entrapment locations unreasonable if the runner and gate design is unreasonable [1].

Cooling analysis. Cooling is another important process after the filling and pressure maintaining. The arrangement of the cooling water has an important influence on the cooling rate and forming

Published by Atlantis Press, Paris, France.

quality of the plastic parts. It is good for shorten the production cycle and reducing the residual stress of plastic parts by using a good cooling solution.

Warping analysis. Warpage was mainly caused by filling imbalance which resulting in high molding pressure and uneven cooling which resulting in uneven contraction [2]. It would affect the assembly of the product if the warpage deformation is too large. If the deformation is larger than the product tolerances, it will cause rejection. The optimize design of filling parameters, maintaining pressure, and the cooling system can significantly improve the product warpage.

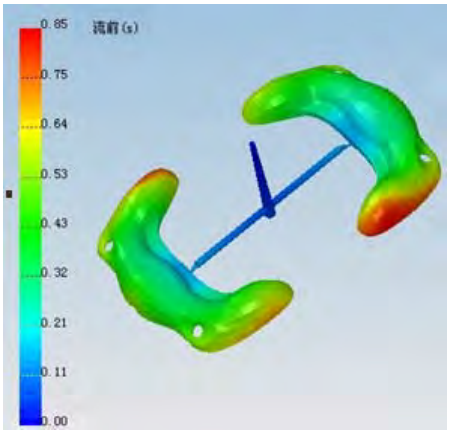
The parameters of parts were shown in Table 1. A completely filled, uniform cooled, small warpage product can be obtained using the injection process parameters in Table 2. The forming results were shown in Fig.2.

Table 1.The parameters of part

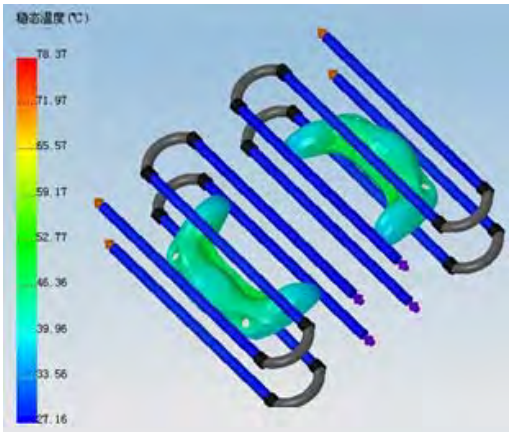
Material	ABS	**Plastic density**	994kg/m^3
Part volume	21353mm^3	**Average thickness**	2.5mm
Runner volume	2993mm^3	**Maximum thickness**	3.2mm
Units number	13044	**Minimum thickness**	1.5mm

Table 2.Injection process parameters

Injection molding machine	100t	**Maximum clamping force**	330KN
Molding temperature	230°C	**Pressure maintaining**	54MPa
Mold temperature	40°C	**Pressure maintaining time**	12s
Maximum injection pressure	61MPa	**Diameter of the cooling water**	8mm
Filling time	0.85s	**Ejection temperature**	79°C
Gate number	2	**Cooling time**	7s

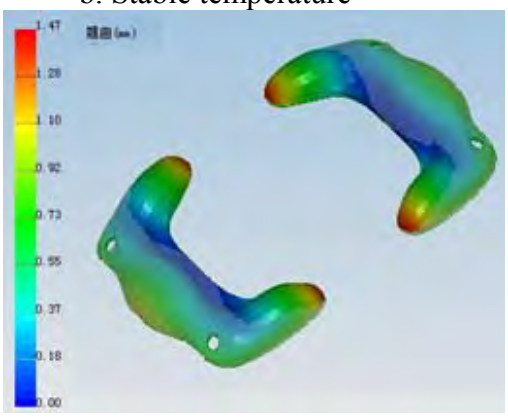


a. Filling time



b. Stable temperature

c. Coolant temperature



d. Warpage deformation

Fig 2.Forming result

CAD mold design

Core and cavity design of mold. A two-cavity mold was adopted, and the parting surface was taken at the maximum dimensions of the plastic part. The material shrinkage of 0.5% was set in the CAD software. The mold core and cavity were cut up after the parting surface was completed. The split core and cavity need the design of cooling water, and the design of the waterway must take the cooling efficiency and cooling uniformity into account [3]. The 3-D graphics of core, cavity and part were shown in Fig.3.

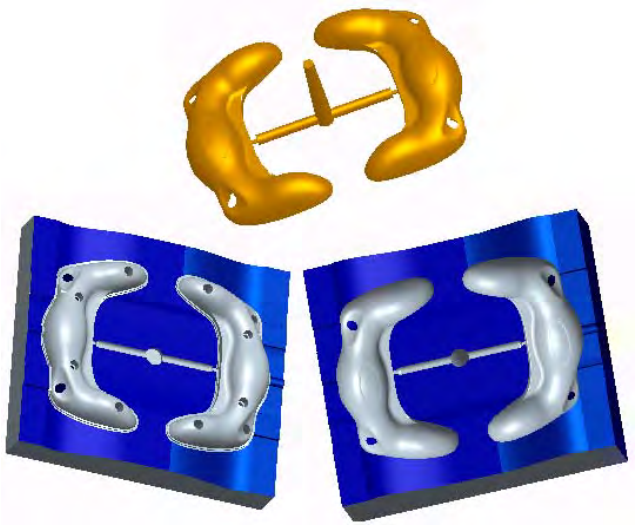


Fig 3.Core, cavity and part

Gate design of plastic part. A submarine gate was adopted because the part did not allow the mark of gate on plastic surface. Many manufacturers adopt the form of submarine gate shown in Fig.4. That was making a channel in an ejector pin as the gate [4]. However, this submarine gate not only made the appearance of bake mark on the part at the ejector pin, but also made the ejector pin wear out easily which produce debris, thus affecting the quality of products. In this paper, the ejector tubes were adopted. In order to avoid the above defects, the hook-type submarine gate were adopted [5], which shown in Fig.5.

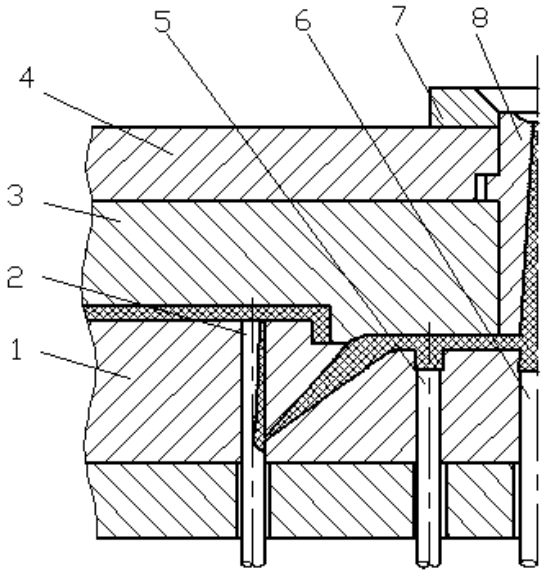


Fig 4. Submarine gate in ejector pin
1-moving plate 2- ejector pin 3- fixed plate 4- fixed base plate 5-ejector pin
6-sprue puller pin 7- locating ring 8-sprue bushing

Published by Atlantis Press, Paris, France.

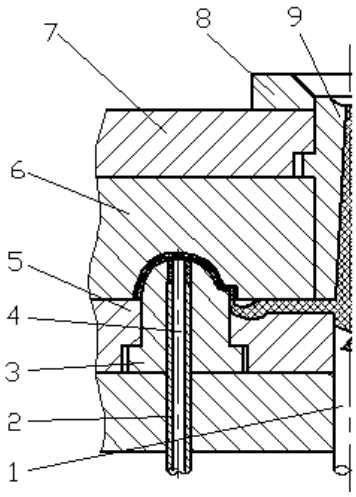


Fig 5.Hook submarine gate

1-sprue puller pin 2-ejector tube 3-punch 4-core 5- moving plate 6-fixed plate 7- fixed base plate 8- locating ring 9-sprue bushing

The overall design of mold. After completing the design of casting system , cooling system, ejecting system and the frame of the mold, the 2-D graphic of the mold was shown in Fig.6. The working process of the mold was as follows: when clamping, the moving mold and fixed mold were closed under the action of the guide pin 9 and the guide bushing 10. The cavity was composed by die, punch and core, and the clamping force was provided by clamping system of the injection molding machine. Then the injection molding machine began to inject, the melted plastic flew into the cavity through the casting system. After the stages of filling, pressure maintaining, filling shrinkage and cooling, the products sizing and then the mold open. When the mold opened, the moving die was

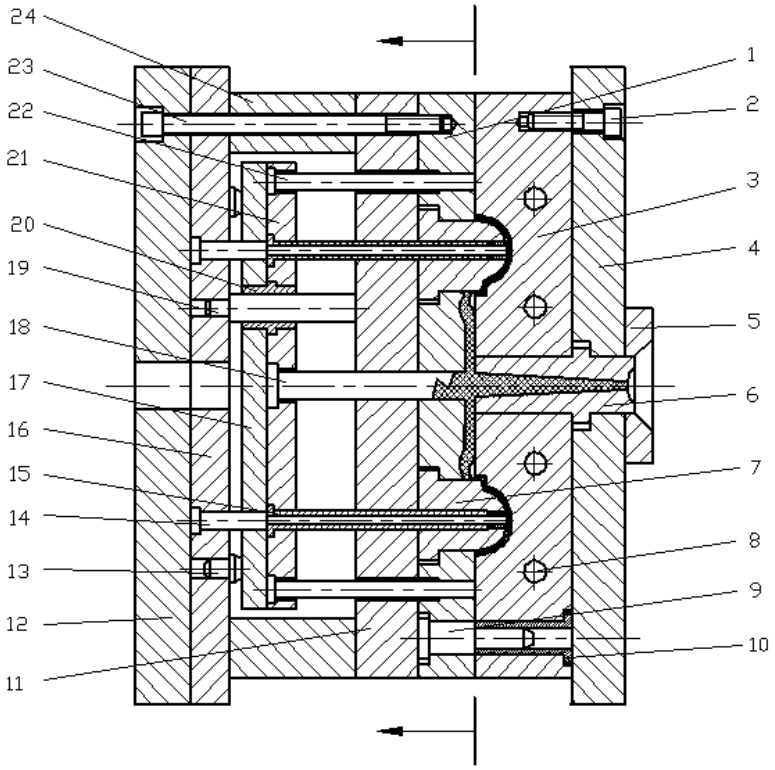


Fig 6.2-D assembly graphic of the mold

1-moving plate 2-bolt 3-fixed plate 4-fixed baseplate 5-locating ring 6-sprue bushing 7-punch 8-cooling water 9-pin 10-guide pin bushing 11-support plate 12-moving base plate 13-support pillar 14-core 15-ejector tube 16-core retainer plate 17-ejector plate 18-sprue puller pin 19-ejector plate guide pin 20-ejector plate guide bushing 21-ejector retainer plate 22-return pin 23-bolt 24-cushion plate

Published by Atlantis Press, Paris, France.

driven back by clamping system of the injection molding machine, and the mold separated from the parting surface. The plastics wrapped on the punchand went back with moving mold. At the same time, the sprue material was pulled out from sprue bushing 6 with pulling pin 18. When the moving mold moved a certain distance, the ejector pin of the injection molding machine through the holes of moving base plate 12 and core retainer plate 16, and then came into contact with the push plate 17. The ejector institutions began to move. The ejector tube 15 ejected the plastic and sprue puller pin push the sprue material which caused the plastic and the sprue material falling from the mold. Then an injection finished.

Conclusions

CAE technology can deal with many data such as product design, manufacturing, analysis, simulation and other data. It can simulate the entire injection molding process before the mold made, including determining the best gate location, filling, pressure maintaining, cooling, warpage, etc [6]. With the combination of mold CAD and CAE technology, it can detect the design defects early, improve part quality, reduce the tryout number, so as to improve productivity and reduce costs.

Acknowledgement

This work has been supported by the project of Dongguan Polytechnic JGXM2011019.

References

[1]L. Guoliang, “Design of Injection Mold for the Electrical Equipment Shell Based on Moldex3D,” Journal of mold manufacturing, vol. 32, pp. 10–17, August 2011.

[2]W. Fan, W. Lishuang, Z. Yuzeng, “Injection plastics process analysis of the plastics decoration parts,” Chinese Journal of new technology and new process, vol. 40, pp. 63–65, February 2011.

[3] L. Guoliang, “Application and Research on Advanced Manufacturing Technology Design Method for Plastic Injection Molding,” Journal of new technology and new process, vol. 34, pp. 44–47, August 2011.

[4] Y. Yehong, L. Qiumei, “Optimization Design of Injection mold for receptacle upper cover based on Pro/E Software,” Chinese Journal of mold manufacturing, vol. 40, pp. 7–10, February 2007.

[5] Y. Zhensheng, “Hook-type submarine gate,” Journal of plastic, vol. 34, pp. 49–50, May 2004.

[6]Z. Qiang, “Plastic products and mold design based on CAE,” Journal of mechanical research and application, vol. 34, pp. 126–128, May 2010.