PaperID:CS355 2022 International Conference on Applied Mathematics, Modeling and Computer Simulation (AMMCS 2022)

Selection and Warpage Deformation Optimization of Clamshell Parts Based on CAE Technology

Guodong Liang
Mechanical and Electrical Engineering Institute,Heyuan Polytechnic, Guangdong 517000

Introduction

We can see that plastic covers, especially flip covers, are an indispensable part of daily necessities. Flip covers have the characteristics of internal threads, hinges, thin walls, small size, and precision. They often encounter molding difficulties in mold development. . At present, many scholars have done a lot of research, but they often focus on one aspect, and the comprehensive research on lids is still very little.

preprocessing analysis

Figure 1 below is the three-dimensional model of the flip cover designed this time. The structure of the plastic part is a bow-knot flip cover with an internal thread structure. The outer diameter of this plastic part is 24mm, and the average wall thickness is less than 2mm. In the range of 0.35-1.90mm, special attention should be paid to its design, and if necessary, reinforcing ribs or bosses should be added. There is a 3.07mm through hole on the plastic part to ensure its strength and size.

Figure 1 . 3D View of the Flip Lid

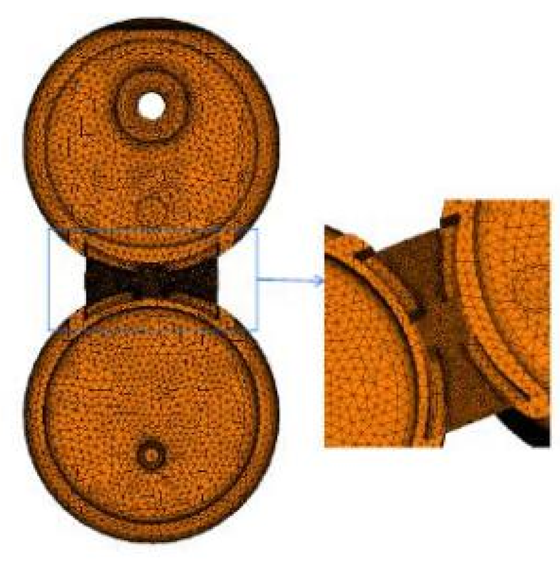


Figure 2. Clamshell meshing diagram

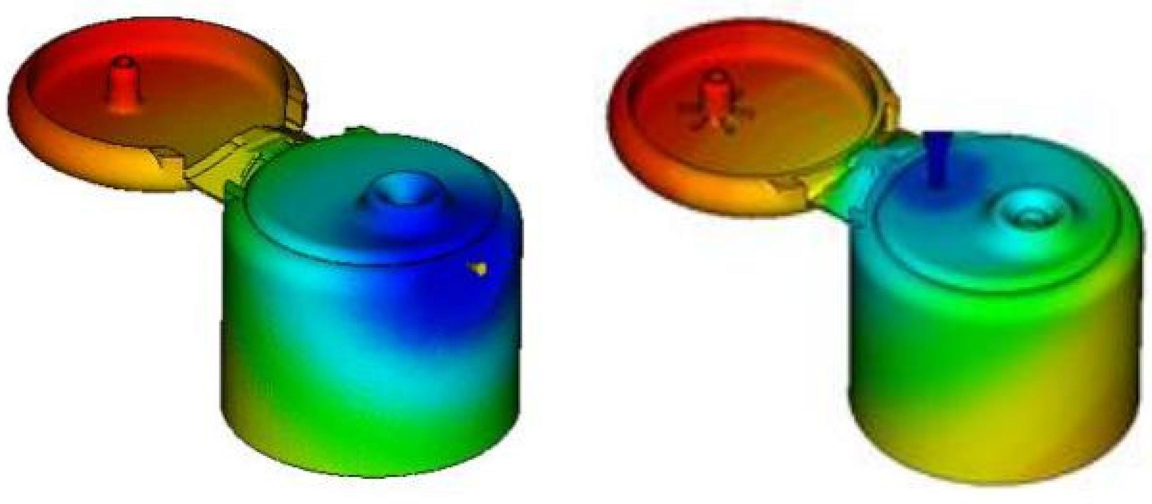
First import the injection molded part model in 3D format, and mesh the product using the double-layer type. The model is divided into a finite number of triangular elements, and at the same time, the elements are connected with nodes, so that the finite elements become a linked whole for analysis. After the mesh division is completed, the mesh needs to be inspected and repaired to ensure that there are no defects such as free edges, so as to prepare for the next analysis. Through initial adjustment, the resulting mesh is divided into 33,008 triangles, and the matching rate of 66,016 nodes reaches 89.1%. No free edges and crossing edges, no misaligned cells and overlapping cells are shown in Figure 2.

Gate location selection

Therefore, the thermoplastic material used is polypropylene (PP), the density is 0.90-0.91cm 3 , and the shrinkage rate is 18-25 ‰.For plastic parts with a relatively complex structure, it is difficult to find the location of the gate by experience. However, in MPI software, its gate analysis module can be used to find a reference optimal gate position through finite element analysis to ensure the balance in the fluid process. At the same time, due to the different applications of cold and hot runners, the gate location is different, so it is necessary to determine a reasonable gate location in combination with the mold structure and product appearance quality . There are currently two gate schemes as shown in Figure 3 below.

(1) Cold runner latent gate, as shown in Figure 3(a).

(2) Hot runner latent gate, as shown in Figure 3(b).



(a) (b)

Figure 3. Schematic diagram of gate scheme

Simulation results

As shown in Figure 4, the blue area is the area filled first, the green area is second, and the red area is the last. It can be seen from the figure that the filling time of the cold runner latent gate is 0.539s, while the filling time of the hot runner needle valve gate is 0.510s. The difference of 0.029s between the two is not very big. Considering that the mold is a mold with 24 cavities in one mold, and the product batch is large , it is still advantageous to consider the hot runner in terms of overall production efficiency, but the cost of the hot runner is high , which needs to be weighed from the size of the product batch . relationship . From the basic symmetrical distribution of colors, it can be seen that both schemes can completely fill the plastic parts without unsaturated mode (short shot). There was also no slight filling imbalance farther from the gate, and both options met our requirements.

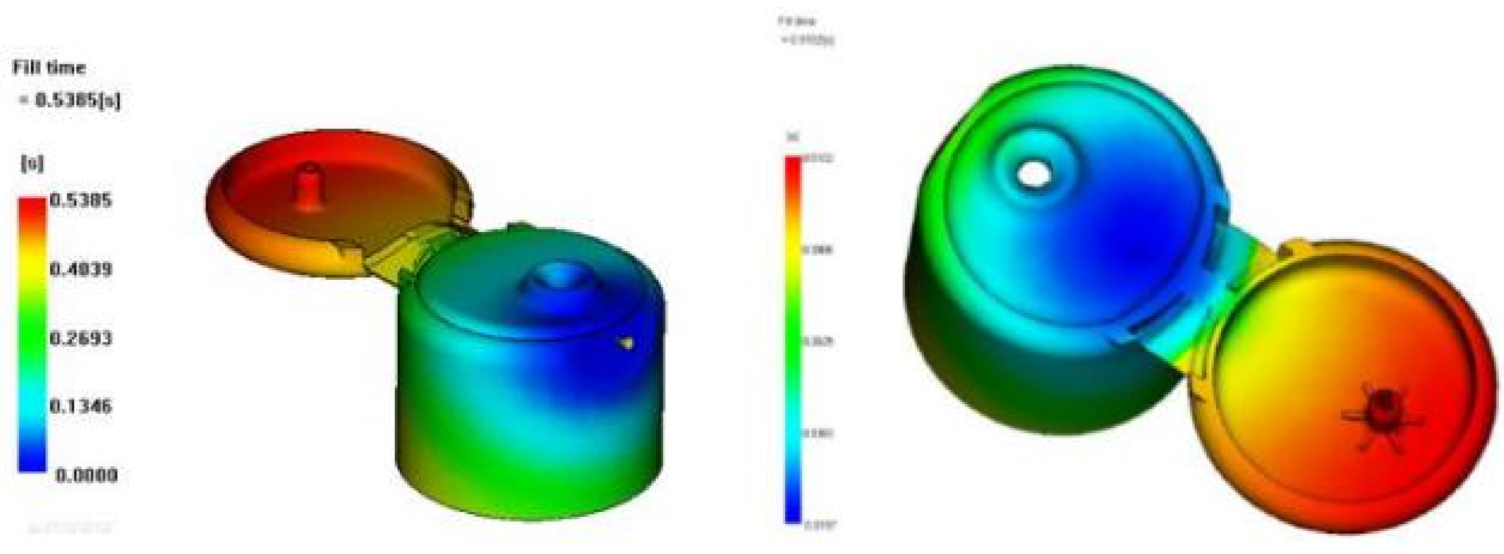


Figure 4. Schematic diagram of filling time simulation (left is cold runner, right is hot runner)

Figure 5 shows the analysis results of the injection mold flow front temperature for two gates. Generally, the temperature of the melt front end needs to be controlled within 300 degrees Celsius . If the temperature change is too severe, there will be residual stress in the plastic part, and the existence of residual stress will cause the part to warp. As shown in Figure 5, the temperature difference between the two types of runners is controlled at about 10 degrees, and the difference is not too big, but if you focus on local areas, you can find that the temperature difference between the cover surface and the cover body of the cold runner latent gate is large. There is a certain risk of warping, while the hot runner needle valve gate has a more uniform temperature distribution, and there is only a small risk of bending at the hinge. There is bending here, so it is still allowed to exist.

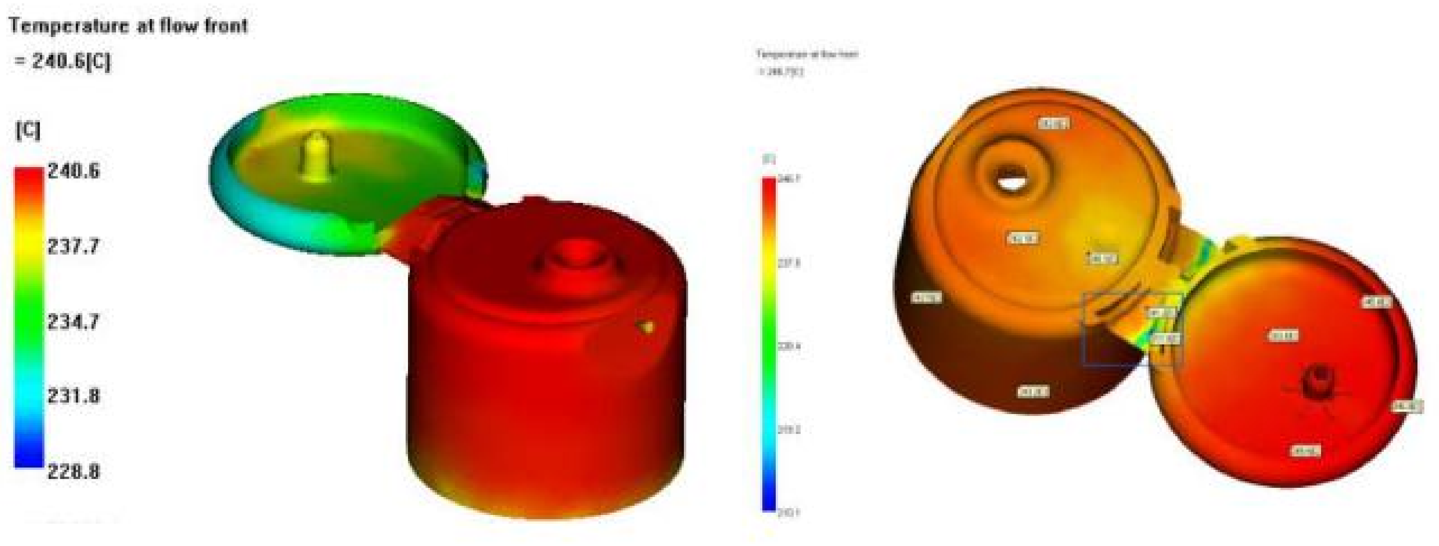


Figure 5. Schematic diagram of temperature simulation of flow front (left is cold runner, right is hot runner)

Through Moldflow simulation, it is found that the place where sink marks are prone to appear is the area shown in red and green in Figure 6. This flip cover has low local rigidity requirements at the sealing plug (marked in green), so the wall thickness has been designed to be relatively thin. Rounded corners are processed. However, sink marks can be seen on the back of the plunger (marked in red) .

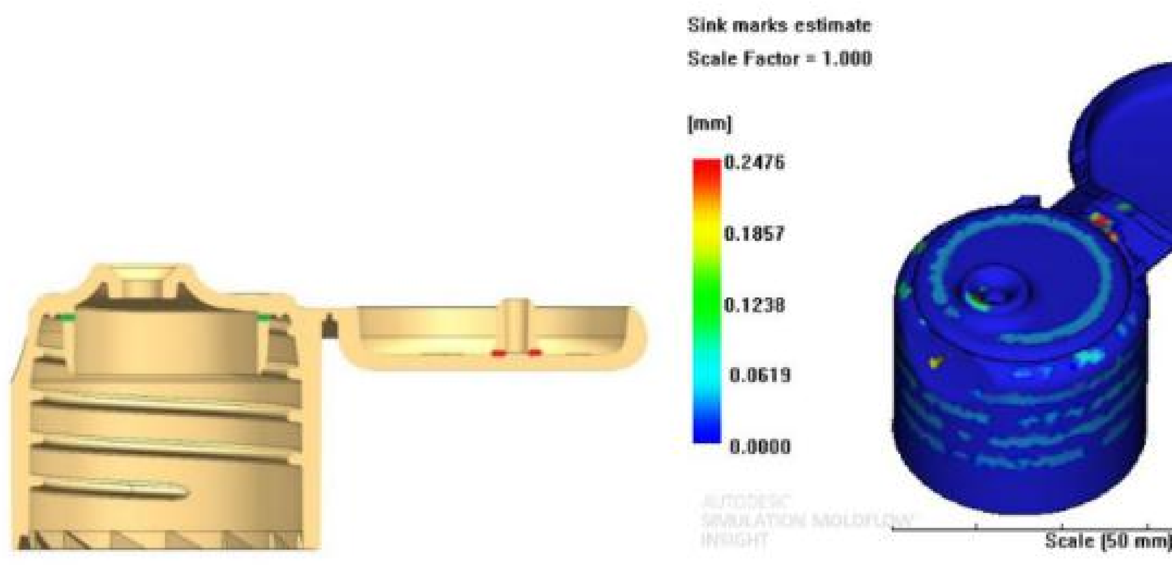


Figure 6 . Simulation of lid sink marks

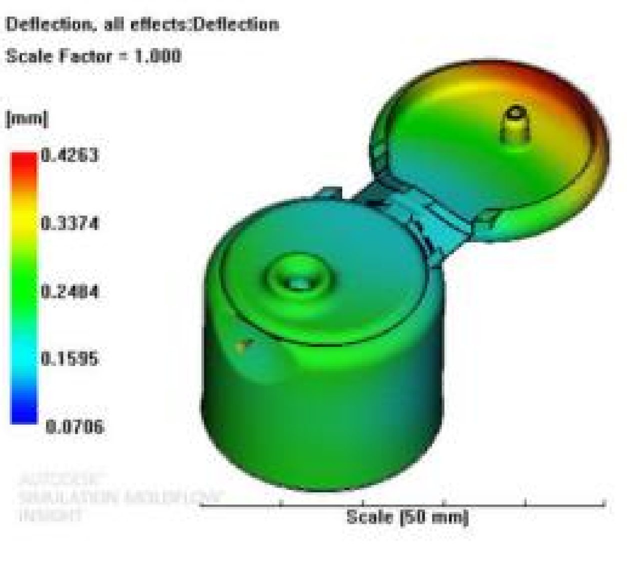


Figure 7. Simulation of lid warpage

After cooling, the plastic parts will also warp to a certain extent. As shown in Figure 7 , the plastic parts are not well cooled during the injection molding process, resulting in different wall thickness shrinkage, which is the main factor of warpage. Moldflow's warpage analysis can accurately predict the possibility of warpage of plastic parts, and provide various possible causes. Warpage analysis is generally performed after flow and cooling have been optimized. The warpage analysis needs to determine the degree of warpage, which is acceptable if it is within the allowable range, otherwise some adjustments must be made to reduce the warpage.

Warping deformation optimization test verification

In order to reduce the magnitude of warpage change of the hot runner scheme, Moldflow is used to optimize the injection parameters and select variable parameters including melt temperature, filling time and filling pressure. The specific experimental results are shown in Table 1 below.From the above analysis, it can be seen that the best combination of parameters for the hot runner process is the melt temperature of 220 ° C, the filling time of 0.5s, and the filling pressure of 96%. At this time, the total warpage deformation is 0.3013, and the molding effect of the part is the best. The optimized parameters are simulated and verified, and the results are shown in Figure 8 below, and it is found that the degree of deformation is narrowed.

Table 1. Warping deformation optimization verification of 15 sets of parameters

Numbering	Factor			Warpage/mm
	Melt temperature/℃	Fill time/s	Filling pressure/%	
1	220	0.4	64	0.4968
2	220	0.6	80	0.3765
3	220	0.5	96	0.3013
4	220	0.5	64	0.3476
5	220	0.4	96	0.4007
6	230	0.6	96	0.3672
7	230	0.5	64	0.3542
8	230	0.4	80	0.4815
9	230	0.4	64	0.3788
10	230	0.6	64	0.3937
11	240	0.5	96	0.4263
12	240	0.4	64	0.3511
13	240	0.6	80	0.3657
14	240	0.5	64	0.5311
15	240	0.5	80	0.5701

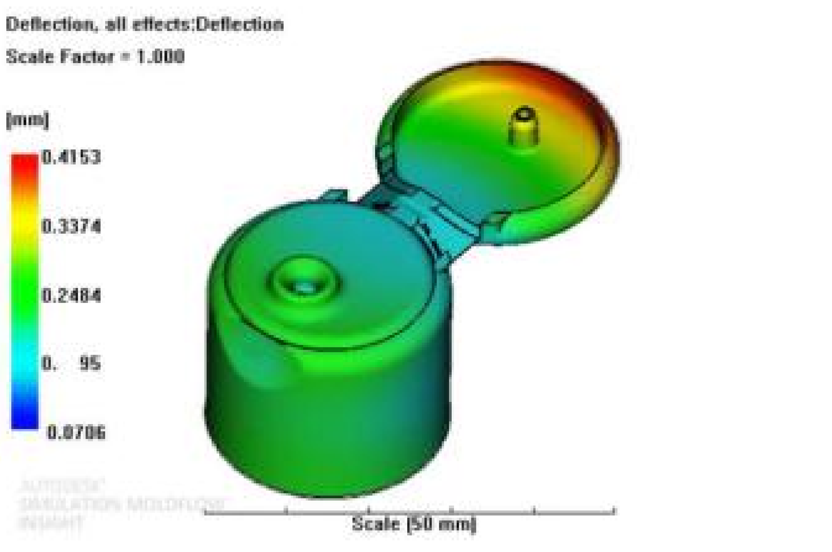


Figure 8. Simulation diagram of warpage deformation of optimal process

Conclusion

All in all, by simulating the process of the flip cover, it can be seen that the hot runner latent gate process has a more excellent molding effect. Then the process parameters were further optimized and verified, and the optimal process parameters were obtained as melt temperature 220° C, filling time 0.5s, and filling pressure 96%. Final injection molding properties of the flip cap.

References

[1] Wang Chang, Zhang Kun, Liu Tonghui. Analysis of injection mold flow of bearing cap based on CAD/CAE technology [J]. Plastic Science and Technology, 2021, 49(3):4.

[2]Chen Wei. Research on three-way injection molding process and mold design [D]. Beijing Jiaotong University, 2019.

[3] Jia Juanjuan, Xu Xiaochang. Design of complex panel injection mold based on mold CAD/CAE technology [J]. Plastics Industry, 2020, 48(6):5.

[4] Dong Kun. Design of injection mold for mobile phone casing based on CAD/CAE and optimization of forming process [D]. Anhui Engineering University, 2016.

[5] Xue Zichuang. Design of injection mold for LED lamp shade based on CAD and CAE technology [J]. Mechanical Research and Application, 2020, 33(4):4.

About the author: Guodong Liang, (1978 -), male, from Guilin, Guangxi, undergraduate, associate professor, mainly engaged in material forming technology and simulation research ; E-mail: 50331624 @qq.com.