DOI: https://doi.org/10.59294/HIUJS.VOL.4.2023.392

Cooling channel design for improving quality of injection plastic product

Pham Tuan Nghia
Tran Dai Nghia University

ABSTRACT

Injection molding is the most widely used technique for making the related plastic parts for consumer electronics goods with limited lifespans, like mobile phones, which are growing more and more popular. The molten plastic beads must be injected into a mold cavity, cooled, then part-formed and ejected. Filling, packing, cooling, and ejection are the four key steps in an injection molding process. The length of the molding cycle affects how economical the procedure is. The cooling step is the most crucial of the three since it controls how quickly are produced. Otherwise, cooling systems are important for the injection molding process in terms of productivity, quality, and mold-making costs. In this paper, three conformal cooling channel designs are proposed for obtaining uniform cooling over the molded parts. The research is conducted by using CAE software (MOLDEX 3D) to simulate the injection process and compare the results of three conformal cooling channel designs with molded part cooling by moldbase. All three designs show that warpage and cycle time improve significantly, helping to decrease cost and increase productivity. In which the combination of a conformal cooling channel outside and a baffle cooling channel significantly reduces the warpage.

Keywords: *Injection molding, plastic product productions, conformal cooling channel (CCC)*

1. INTRODUCTION

Nowadays, plastic products appear in many segments of life such as electronics equipment, healthcare, construction, transportation, agriculture, wide range of use include outstanding material characteristics like high strength-to-weight ratio, stiffness and toughness, ductility and low lifetime cost. Among many ways to produce plastic products, the injection molding process seems to be the most common method.

The basic principle of injection molding is to melt plastic pellets and inject them into a cavity in a mold. After cooling, the part is ejected from the machine. Therefore, the main stages of the injection molding process are filling packaging, cooling and ejection. Process economics depend on the time spent in the molding cycle. The cooling phase accounts for up to 80% of the total cycle time and directly influences the shape deviations (due to shrinkage, bending, warping, etc.) of the final plastic part [1] , so it determines the rate at which the parts are produced. Therefore, the cooldown phase is the the most important step of three.

Corresponding Author: Pham Tuan Nghia
*Email: nghiapt91@gmail.*com

Cooling systems, which typically consist of a succession of straight-drilled holes as shown in Figure 1 [2], are required in molds in order to cause the injected materials to cool down. Straight-drilled channels cannot provide ideal cooling because their layouts are constrained by the cavity form (to minimize interference between the cavity and channels) and drilling technique (only straight holes may be drilled), despite the fact that they are simple and inexpensive to make. As a result, straight-drilled circular channels are required by manufacturing limitations even if they might not be the best choice for the mold cooling process.

In recent years, the cooling efficiency has not been as high as desired when using straight-drilled cooling molds. On the other hand, sharp turns at the connection of two adjacent straight-drilled channels (Figure 1), limit the coolant mobility, causing a rapid pressure drop that reduces the cooling capacity downstream and increases the uneven cooling[3].

The parts created via thermoforming typically have a thin shell, intricately curved surfaces, and/or deep hollow structures [4]. Due to the deep hollow feature's propensity to cause localized heat accumulation, differential shrinkage and warpage are more severe and cooling rates are substantially lower in these circumstances [5]. This may necessitate costly mold correction to guarantee that the part has the desired accurate dimensions [6]. Conformal cooling (CC) channels were introduced in the 1990s as a way to reduce warpage and accelerate cooling. The CC channel has a constant distance from the cavity surface and is designed as a curved channel[7]. This reduces the distance between the channel and the cavity surface and guarantees a consistent cooling rate along the channel. CC channels can therefore greatly improve the cooling performance of the mold by minimizing temperature differentials[8].

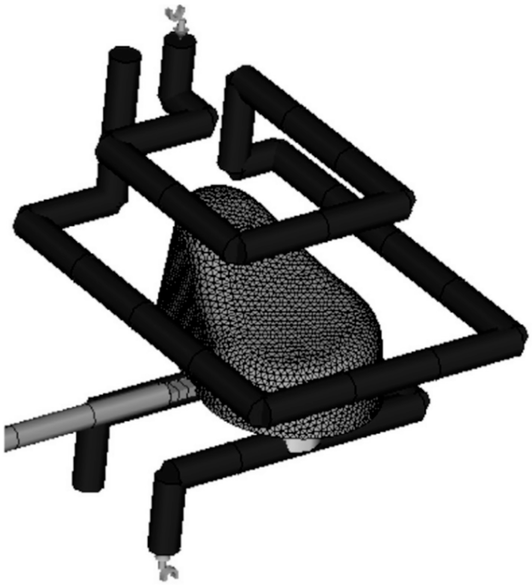


Figure 1. Conventional cooling channel (straight-drilled holes) in mold [2]

In some cases, with proper cooling channel designs, CC channels can reduce the cooling time by as much as 80% [1] and the cycle time by 60 – 70% [9]. Thus, it is clear that a CC channel network's ideal design is essential for producing products more rapidly, consistently, and effectively. To achieve specifications for cooling performance, mechanical strength, coolant fluidity, and other factors, several fundamental design criteria must be followed.

The demand for using the trigger spray head is increasing today. It is easy observe that in the Covid-19 the sanitizer spray bottles (Figure 2a) are very common and useful in epidemic prevention. So that is a challenge to the manufacturing system about parts production of trigger spray heads (Figure 2b).

a. The sanitizer spray bottles | b. The trigger spray head

Figure 2. The sanitizer spray bottles are used in Covid padamic prevention

On the other hand, all of the trigger spray head parts are products of the plastic industry, and some defects appear in components of the trigger spray head such as the head cover. It can be seen in Figure 3 that the bottom dimension is smaller than the upper one, possibly due to warping. In this paper, the cooling channel designs of the head cover and are investigated in order to improve the quality of injection products by reducing warping (deformation) due to shrinkage and increasing productivity by reducing cooling time.

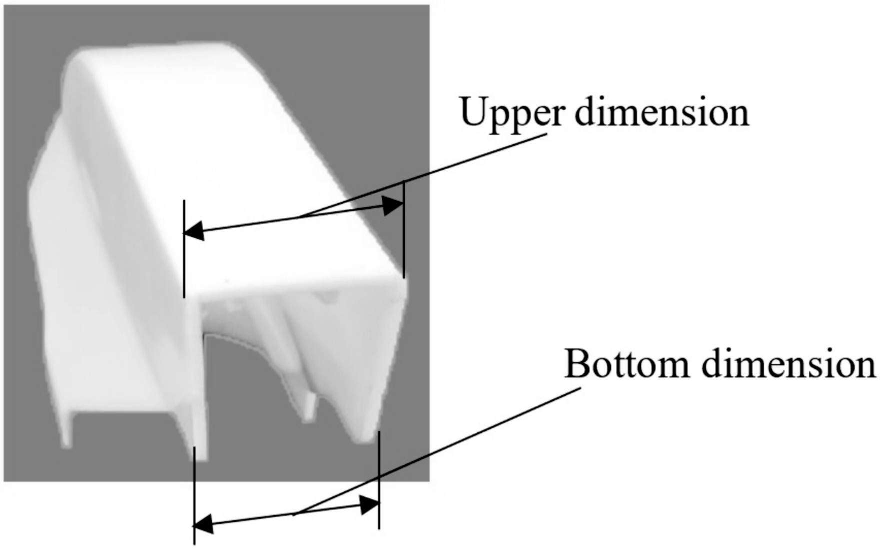


Figure 3. The head cover image is taken from the real product

2. MOLDED PART ANALYSIS

2.1. The model

The real part has been redesigned using CAD software and its primary dimensions are shown in Figure 4.

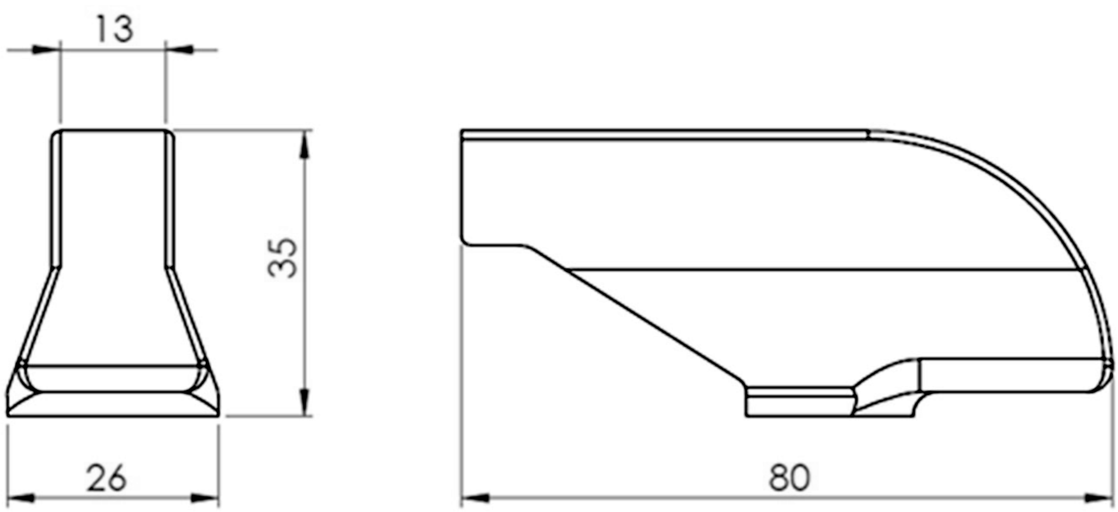


Figure 4. The primary dimension of head cover

2.2. Gate location

The injection gate can only be positioned on the product's two sides due to its geometry; if it were to be placed on the top or bottom, it would prevent the mold from being separated after each injection. Additionally, the gate needs to be on the other side of the product because one side has a small thickness (1 mm). Even so, the placement of the gate guarantees that the ratio L/t stays within the acceptable range for PP materials. (Figure 5).

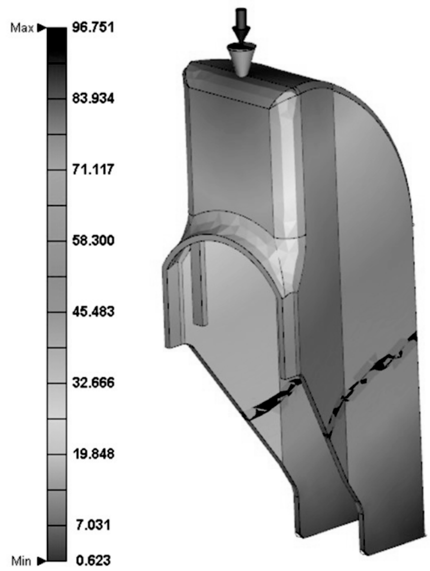


Figure 5. Gate location and L/t ratio

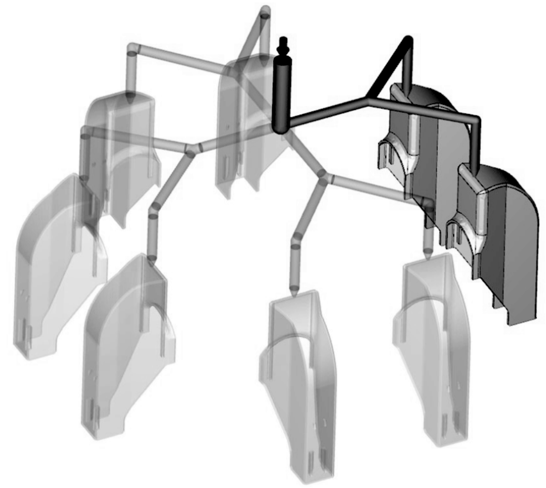


Figure 6. Runner configuration type symmetry 1/4

2.3. Runner system configuration

Due to the relatively small size of the details, the multi-component molding function should be used, specifically employing a hot runner system (Figure 6). In this project, a hot runner system is utilized, where the main runner has a diameter of 8mm and the sub-runner has a diameter of 4.5mm.

2.4. The trial simulation

In the trial simulation, the molded parts use a moldbase (Figure 7) without a cooling channel (parts are cooled by the moldbase) for the purpose of investigating the main factors causing defects for the part, thereby correcting the parameters of the injection process and serving as a basis for designing the proper cooling channel design. The moldbase dimension is 320x320x310mm (LxWxH).

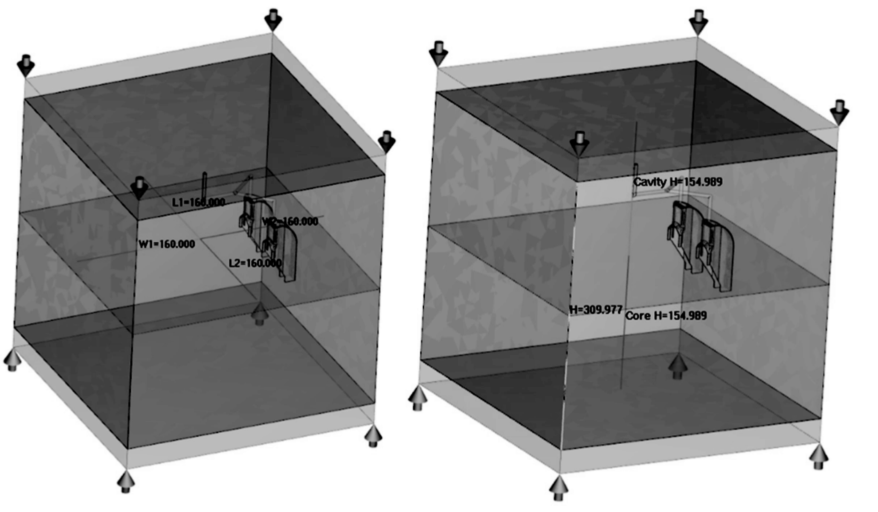


Figure 7. Moldbase without cooling channel

After the trial simulation, the parts are completely filled in after 1.286s and no weldlines appear, the molten core is completely frozen after 4.575s at the end of the packing process. The cooling phase takes 3.701s for the parts to reach the ejection temperature. The sinkmark displacement and the volumetric shrinkage are quite small, as shown in Figure 7. The total displacement is maximum at the end of the part as shown in Figure 8a and the maximum value is 0.868mm the differential shrinkage effect on the displacement is significantly greater than the differential tempe-rature (Figure 8b). Base on the result of trial simulation, main reason cause the warpage that is the differential shrinkage. However, the volumetric shrinkage is quite low and the molten core is completely frozen, so the packing pressure and the packing time are efficient for this stage.

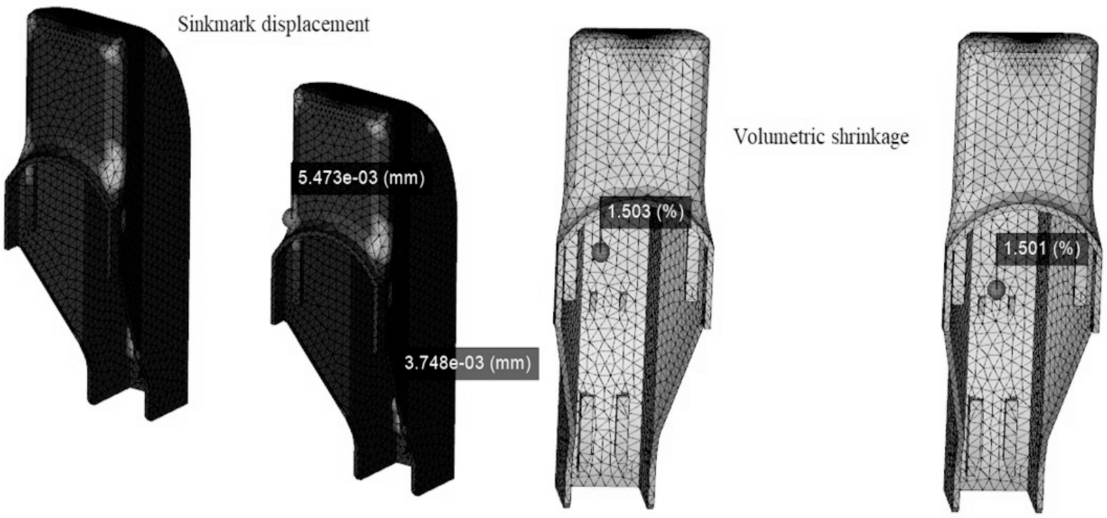
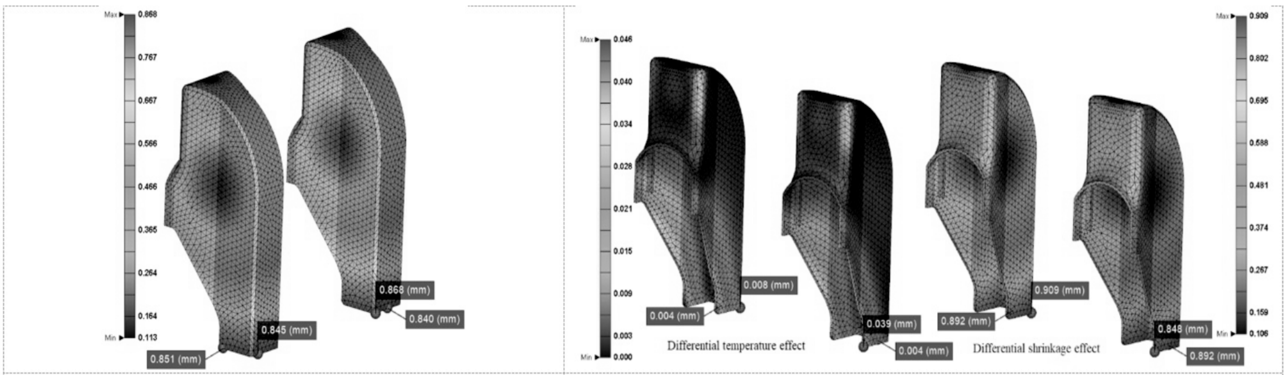


Figure 8. The sink mark displacement and the volumetric shrinkage

When checking the temperature distribution at the EOP (end of packing), the differential temperature between inside and outside of the part, the top and bottom of the part, as shown in Figure 9, is about 20°C (to be considered). This differential makes the shrinkage uneven. Therefore, the need for a cooling channel inside the part is verified.



a. The total displacement at the node of parts | b. Differential temperature and shrinkage affect to displacement

Figure 9. The total displacement at the node of parts

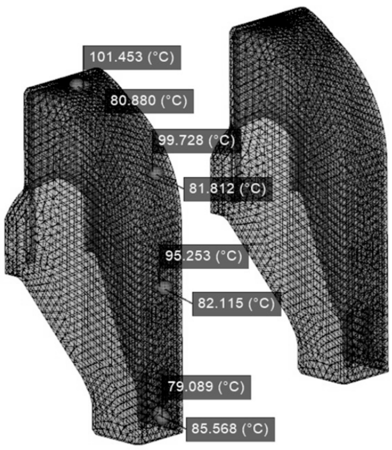


Figure 10. The temperature distribution at EOP

2.5. Cooling channel design

For cooling the part, the designed cooling channel has three configurations:

1. The baffle cooling channel (Internal cooling to reduce the differential of temperature between the inside and outside of the part) (Figure 11a)

2. The conformal cooling channel (CCC) inside is designed with the shape as can be seen in Figure 11b so that the CCC's path follows the internal geometry of the part.

3.The combination between the baffle cooling channel and the conformal cooling channel outside (Figure 11c).

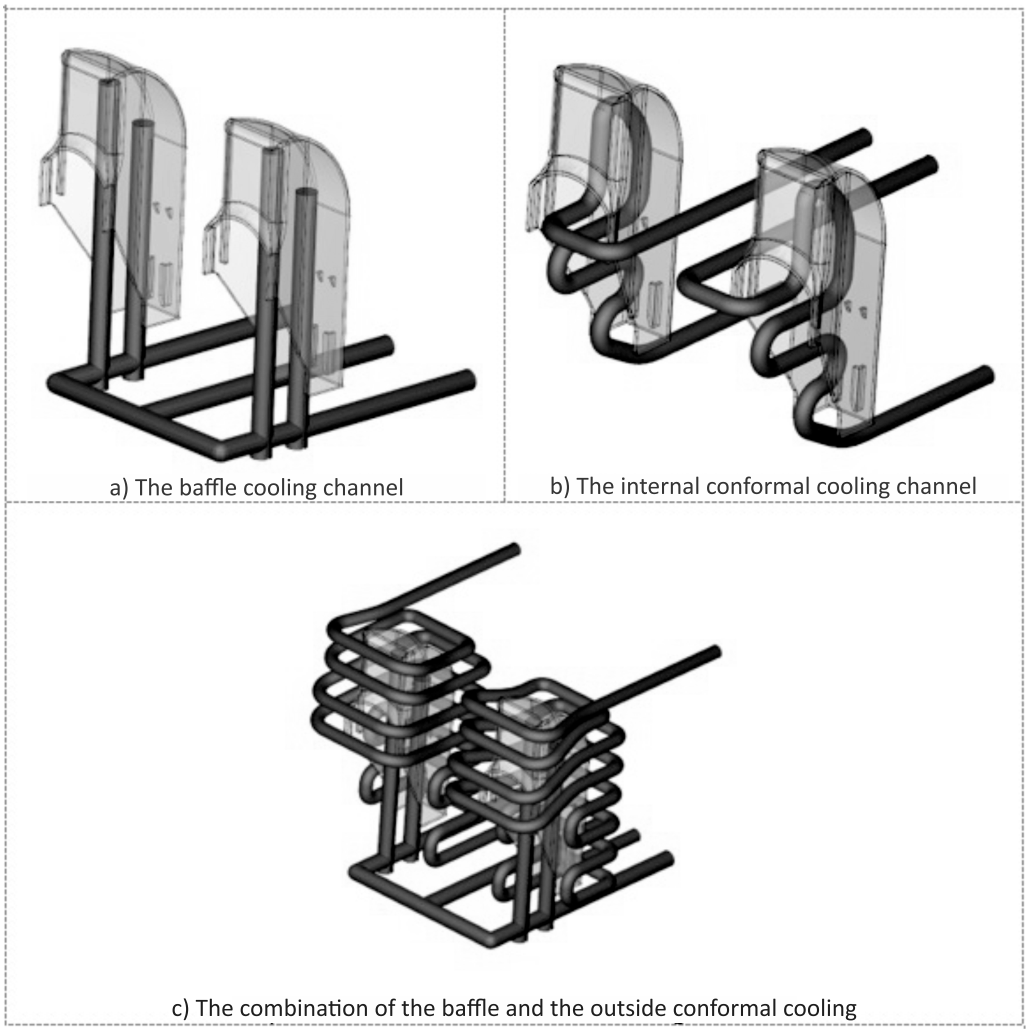


Figure 11. Three kinds of designed cooling channel

3. RESULTS

The temperature difference between the inside and outside elements, as well as between the upper and lower parts of the part, is significantly reduced when cooling channels (Figure 12) are used. In which case, the combined design case shows the most effectiveness in reducing this temperature difference (the difference in temperature is only about 1.1°C) as can be seen in Figure 12c.

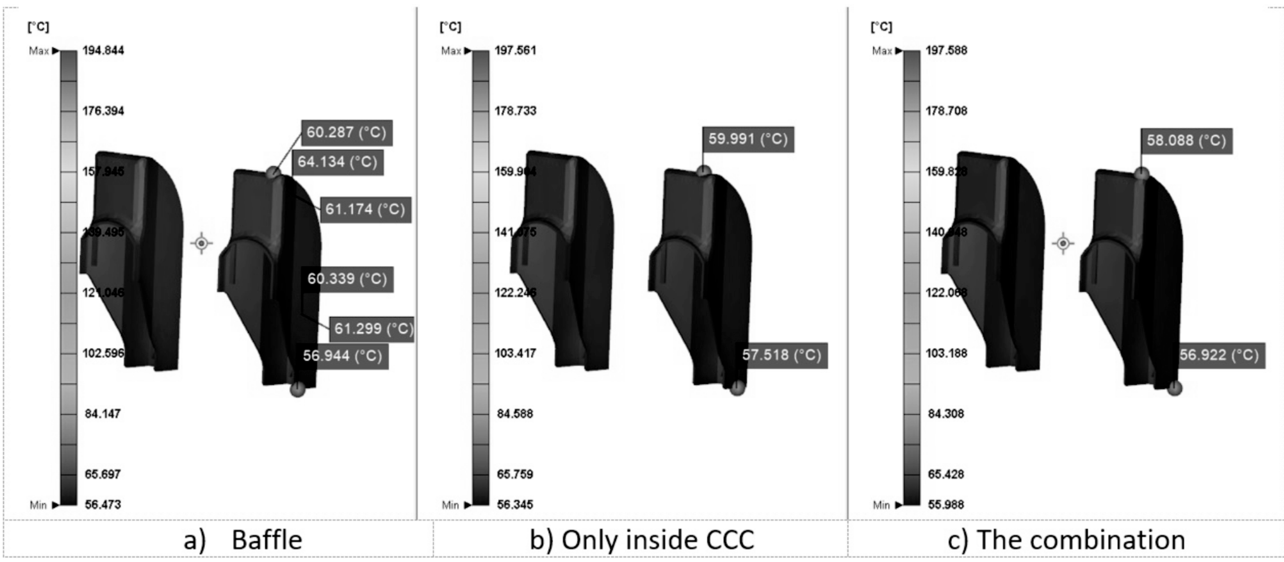


Figure 12. The temperature distributrion of 3 cases at EOP

In comparison to the cooling of the mold, the cooling time takes around 78% less time (about 0.8 sec to reach the ejection temperature), as shown in Figure 13 and in all three cases was constructed, the baffle layout provided the fastest cooling time. It is likely that this case had the least flow length since the cooling system.

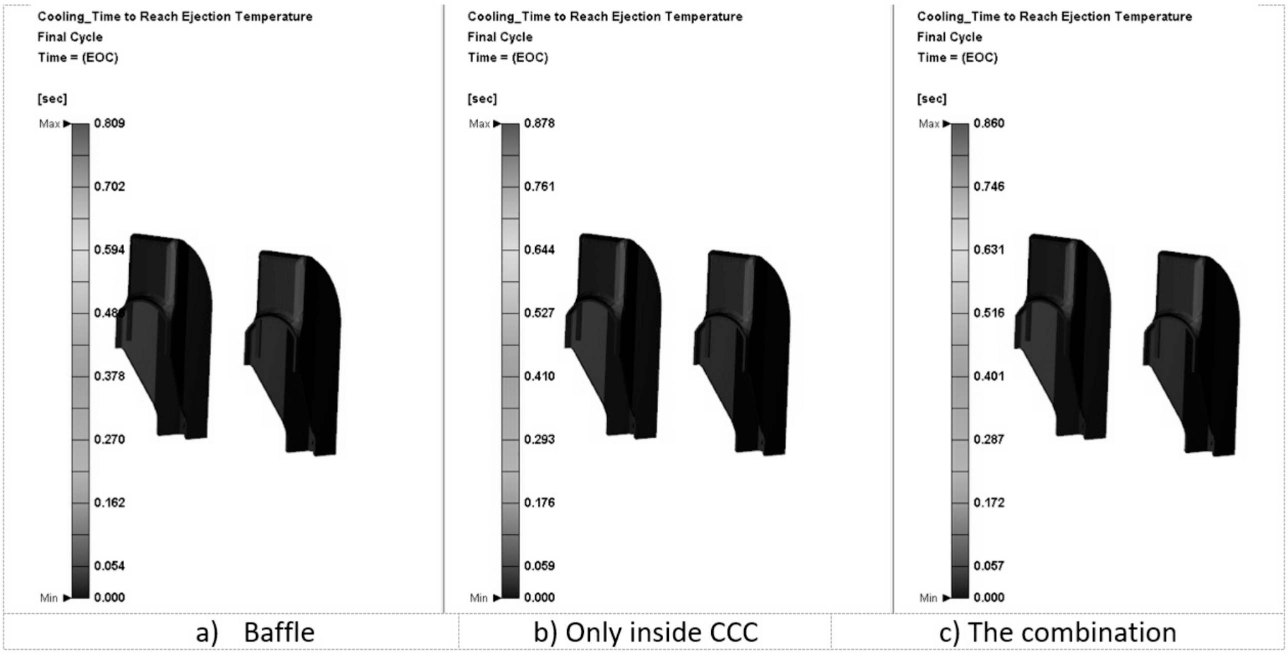


Figure 13. Time to reach the ejection time of three cases

As the temperature difference becomes uniform, the volume shrinkage is reduced. As can be seen in Figure 14 the figure, the warping improves significantly (75% reduction compared to not using the cooling system). In particular, the combined design configuration reduces the most warping (only 0.215mm at the point where the most warping occurs).

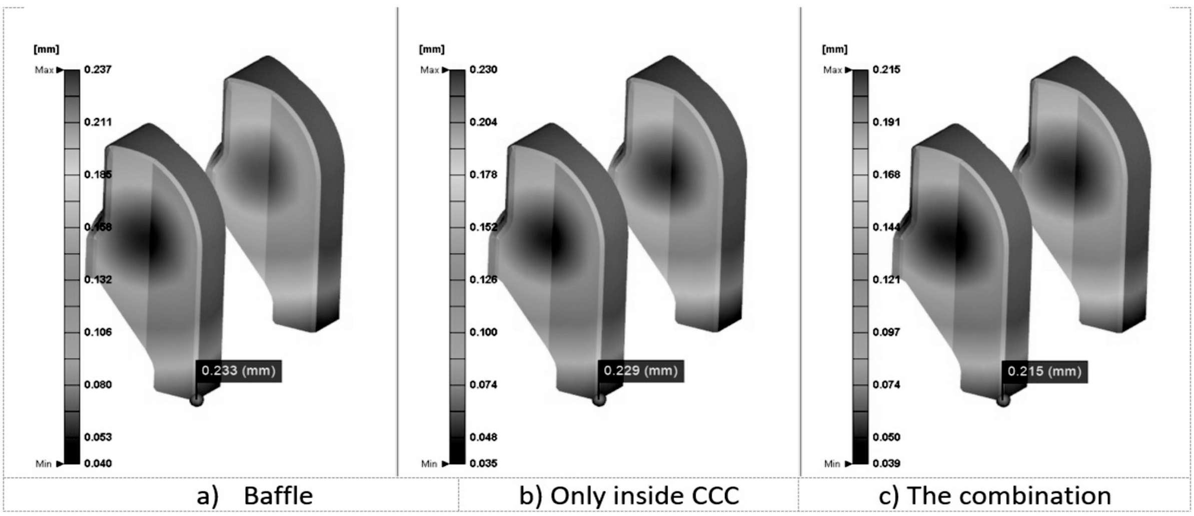


Figure 14. The total displacement of three cases

4. CONCLUSION

In this research, designs of cooling channels for injection-molded plastic parts have been proposed. The results show that using the proper cooling channel will significantly reduce the cycle time, leading to increased productivity and helping the product's cost to improve competitiveness. The article also shows that the use of CAE software in simulating the pressing process of the product saves time and money for the trial production process. Besides, it helps to optimize process parameters and adjust the design to significantly improve product quality and cost.

REFERENCES

[1] N. T. S.H. Masood, "Thermal analysis of conformal cooling channels in," *in: Proc. 3rd BSME-ASME Int. Conf. Therm. Eng, Dhaka*, 2006.

[2] R. E. J. Meckley, "A study on the design and effectiveness of conformal," *Technol. Interface J. 10*, 2009.

[3] A. B. Saifullah and S. H. Masood, "Finite element thermal analysis of conformal cooling channels in injection moulding," *in: Proc. 3rd BSME-ASME Int. Conf. Therm. Eng*, 2006.

[4] M. C. F. M. D.E. Dimla, "Design and optimisation of conformal cooling channels in injection moulding tools," *J. Mater. Process. Technol*, 2006.

[5] K. L. A. N. Y.F. Sun, "Design and FEM analysis of the milled groove insert method for cooling of plastic injection moulds," *Int. J. Adv. Manuf. Technol*. 24, 2004.

[6] L. P. F. S. F. R.-O. E. Mathey, "Automatic optimization of the cooling of injection mold based on the boundary element method," *AIP Conference Proceedings*, 2004.

[7] X. D. H.S. Park, "Optimization of conformal cooling channels with array of baffles for plastic injection mold," *Int. J. Precis. Eng. Manuf*. 11, 2010.

[8] K. W. Kunnayut Eiamsa-ard, "Conformal bubbler cooling for molds by metal deposition process," *Computer-Aided Design*, vol. 69, pp. 126-133, 2015.

[9] W. S. J. D. K. F. D. A. V. B. T. Evens, "Experimental analysis of conformal cooling in SLM produced injection moulds: Effects on process and product quality," *AIP Conference Proceedings 22*, 2019.

Thiết kế kênh làm mát nâng cao chất lượng sản phẩm nhựa ép phun

Phạm Tuấn Nghĩa

TÓM TẮT

Công nghệ ép phun là kỹ thuật được sử dụng rộng rãi nhất để tạo ra các sản phẩm nhựa liên quan đến các sản phẩm điện tử tiêu dùng có tuổi thọ giới hạn, như điện thoại di động, đang ngày càng trở nên phổ biến hơn. Nhựa nóng chảy phải được ép vào một khuôn đúc, làm mát, sau đó chi tiết hình thành và được đẩy ra. Điền đầy, bão áp, làm mát và đẩy ra là các bước chính trong quy trình ép phun. Độ dài của chu kỳ ép phun ảnh hưởng đến tính kinh tế của quy trình. Giai đoạn làm mát là quan trọng nhất trong bốn bước này vì nó kiểm soát tốc độ sản xuất các bộ phận. Mặt khác về phương diện năng suất, chất lượng và chi phí làm khuôn, hệ thống làm mát là rất quan trọng đối với quy trình ép phun. Trong bài báo này, ba thiết kế kênh làm mát phù hợp được đề xuất để đạt được sự làm mát đồng đều trên sản phẩm ép phun. Nghiên cứu được tiến hành bằng việc sử dụng phần mềm CAE (MOLDEX 3D) để mô phỏng quá trình ép phun và so sánh kết quả của ba thiết kế kênh làm mát với quá trình làm mát của sản phẩm ép phun bằng khuôn đúc. Kết quả mô phỏng của cả ba thiết kết kế đều giúp cải thiện đáng kể độ cong vênh của sản phẩm và giảm thời gian chu kỳ ép, giúp giảm chi phí và tăng năng suất. Trong đó, thiết kế kết hợp kênh làm mát bên ngoài và bên trong cho hiệu quả nhất về giảm độ cong vênh của sản phẩm ép.

Từ khóa: *Công nghệ ép phun, sản xuất sản phẩm nhựa, kênh làm mát phù hợp*

Received: 16/05/2023
Revised: 06/06/2023
Accepted for publication: 08/06/2023