ISSN: 2456-3676

Improve Product Design and Injection Molding for Controler Plate Plastic with the Help of CAE Software

Vuong Thi Nhu Yen[1], Nguyen Manh Ha[2]

[1]College of Economics and Techniques
Group 8, Thinh Dan ward, Thai Nguyen City, Viet Nam

[2]College of Economics and Techniques
Group 8, Thinh Dan ward, Thai Nguyen City, Viet Nam

doi: 10.51505/ijaemr.2023.8508 URL: http://dx.doi.org/10.51505/ijaemr.2023.8508

Received: Sep 05, 2023 Accepted: Sep 11, 2023 Online Published: Sep 19, 2023

Abstract

CAE (Computer-Aided Engineering) analysis is based on the sequence characteristics of the system, combining model reasoning to conduct analysis, the results have physical significance, is Know-Why but not the Know-How of traditional experience, thus it is possible to systematize and scientificize the injection molding parameter and design types for the order of state and product quality, achieving the goal of scientific injection molding (Scientific Molding). Due to the reliability of CAE results, it is possible to point out potential problems during injection molding and design, propose design modifications and possible solutions to obstacles and possible solutions, can avoid unsafed experience.

CAE at the design stage can be performed on a computer for design modification options to conduct evaluation (Evaluate), evaluate (Verify) and optimize (Optimize), reducing time and cost mold testing, actual mold fixing, shortening actual trial and error cycle, shortening product development time and time-to-market, reducing waste, time and cost in stages. CAE can help injection molders predict and capture injection molding parameters for product quality impact, find out the processing direction (Processing Window) and optimize injection molding parameters. CAE can indicate the main factors affecting the quality of injection molding, thereby providing design modification parameters, injection molding parameters and quantitative criteria.

In this article applied the CAE method to improve injection molding added the the best product. There are a lot of results: First of all, when making the mold, we will make the inserts to escape the air at the position of the cylinders. And the air outlet at the edge of the product so that the air at the edge can escape well; After improving the gate position, cooling water line, pressure protection, mold temperature, we see that the product has a change in warping, but it is not significant.

Keywords: CAE, Evaluate, Verify, Optimize, Injection molding.

Introduction

Analyzing the process flow in the injection molding process can help detect unreasonable points early in the product design process, code design as well as make useful predictions about the product or optimize injection molding conditions. However, for many reasons, this job is still not important in Vietnam and most of our engineers rely heavily on experience. I'm not saying experience is not important, but when products have increasingly shorter life cycles, higher complexity and increasingly fiercer competition, relying on experience alone is not enough.

If you work in the field of mold design and plastic product manufacturing using injection molding technology, you have certainly encountered the following problems:

The customer requested a preliminary flow analysis before designing the mold.

Assume that the product design is not good and you can improve it.

Doubts about the balance of flow or the optimality of the plastic channel system.

There are many cooling options, but I don't know which one is more effective.

The pressed product is defective but I don't know what is the main reason.

With many years of experience providing flow analysis solutions on Moldex3D software for many large businesses, I am confident to bring you analysis solutions that fully meet your requirements. include Analyze and evaluate the technological capabilities of the product (structure, thickness, materials) Analyze and simulate the entire pressing process to find reasonable mold design solutions Analyze product defects based on mold data and pressing technology parameters to improve molds.

Method and results

1. Product shape and size

Technical requirements: Products need to ensure relatively high accuracy, warping according to technical standards, durability, impact resistance and good heat resistance. Plastic materials are safe for users. Aesthetics: Limit defects (weld seams, burns, not filled, ...), avoid the appearance of spray port marks and stains on the front of the product.

Product name: Controller Plate

SKU: D4TD0

Product size: 434.4x91.16x30 (mm)

Product features: is a product that covers the control panel inside the washing machine. No high requirements on appearance, and size.

2. Product material, mass and volume

Material: Plastic; Material code: Lupol HI 4352; Manufacturer: LG Chemical; Density: D = 1.17 g/cm^3; Melting temperature: max = 240℃, min = 210℃, average = 225℃

Check product volume after design by NX software: Product weight: 122.19 g; Product volume: 104439.37 mm^3.

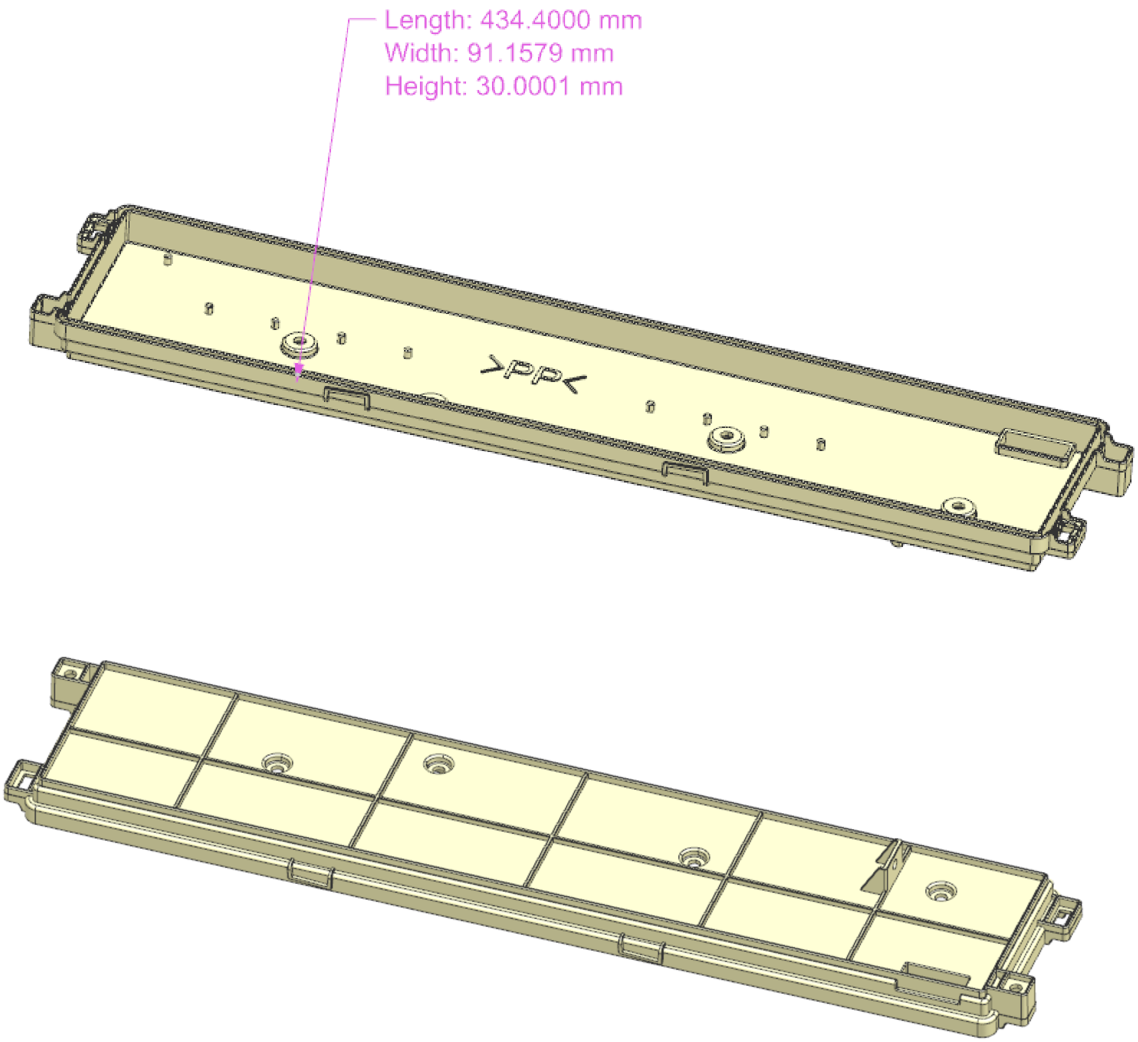


Figure 1. 3D of Product

3. Product thickness

Check product thickness based on design in NX software.

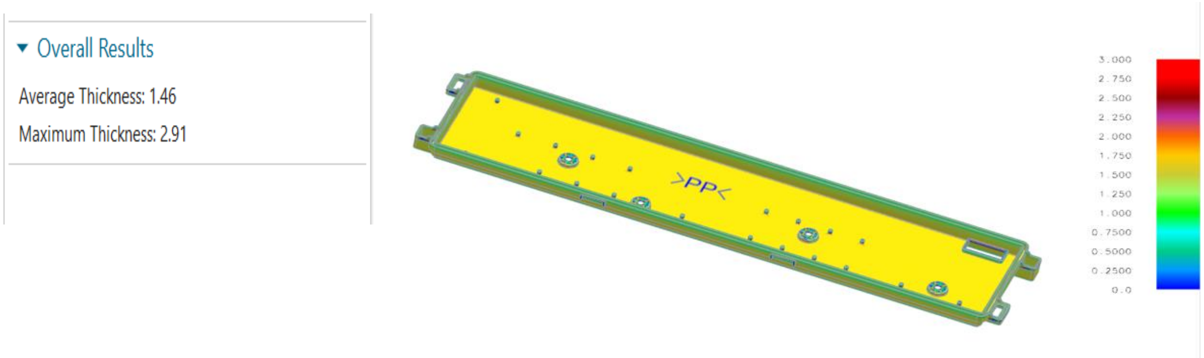


Figure 2. Check product thickness with NX software

Maximum thickness of the product: 2.91 mm; Average thickness of the product: 1.46 mm. Satisfy the usual thickness: 0.5 – 4 mm

Meaning of product thickness inspection: Product thickness directly affects the rigidity, electrical insulation, heat insulation and aesthetics of the product.

Reasonable product thickness design will avoid product errors, increase filling time and shorten injection molding cycle as well as mold making time. Thereby reducing the cost of products and molds, and at the same time saving plastic materials while still bringing efficiency to the product, avoiding defects in injection molding.

The product thickness check is convenient for simulating the injection molding system (affecting product defects), and at the same time letting the designer know whether the product meets the design requirements or not as well as know whether the product meets the durability condition or not.

4. CAE finds the best injection port location

Analysis of the best plastic inlet location from MOLDEX3D 2022

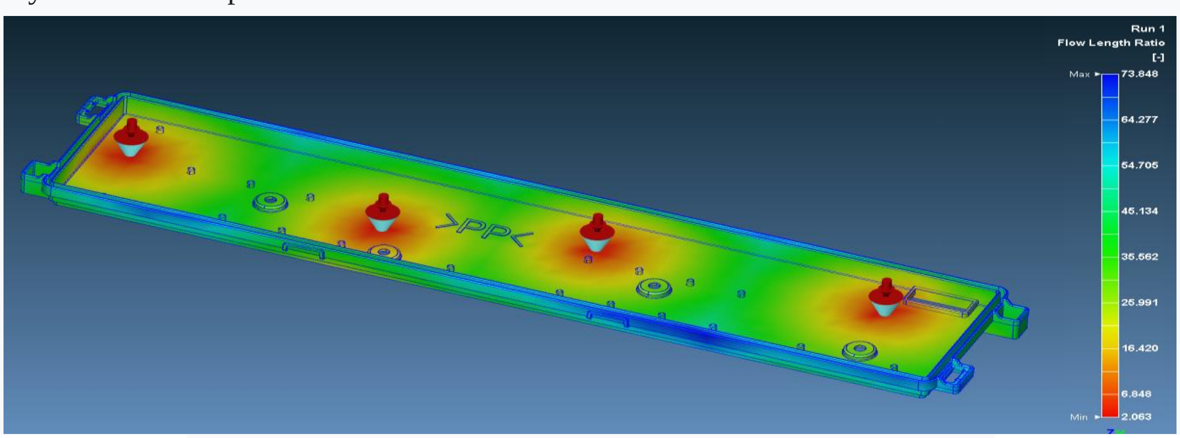


Figure 3. Spray port placement as recommended by Moldex3D

Vol. 8, No. 05; 2023

ISSN: 2456-3676

5. Select the injection port and CAE position to check the weld line and the air bubble concentration position.

Select the injection port position as the original design for analysis

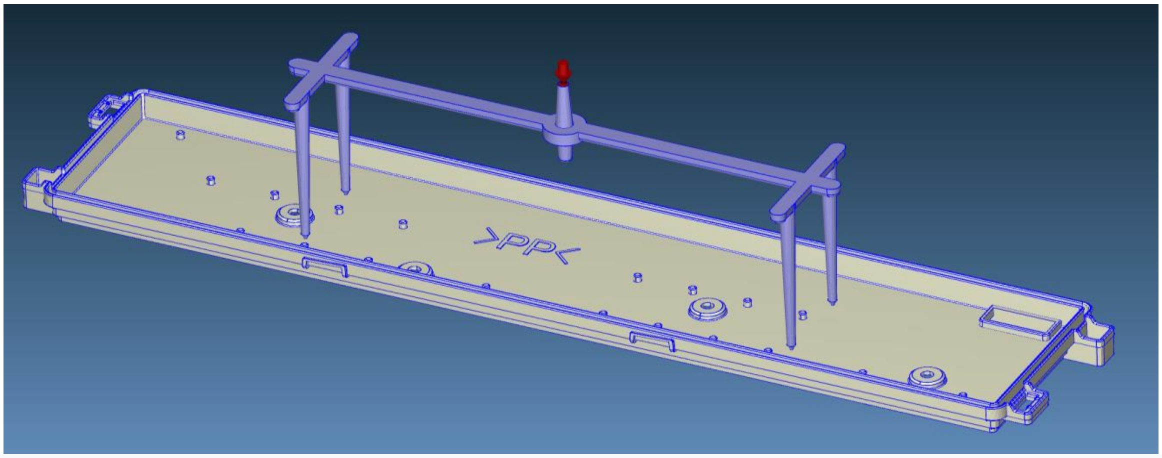


Figure 4. The position of the spray port according to the original design

5.1 . Welding line

Welding simulation results.

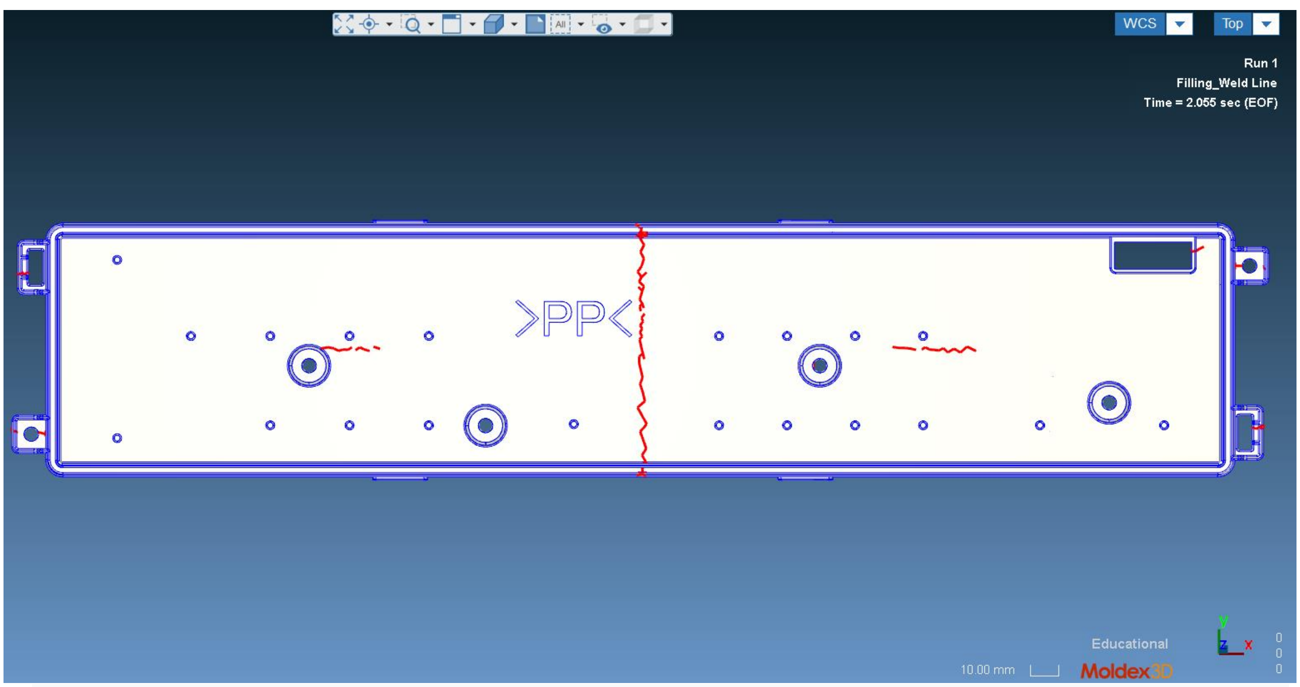


Figure 5. In front of product

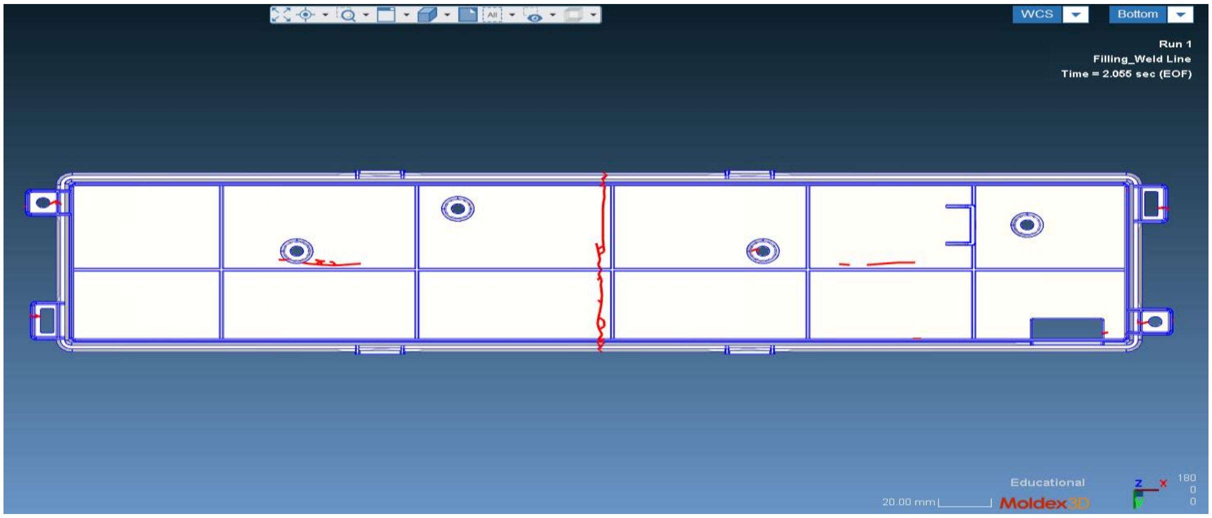


Figure 6. Behind of product

Vol. 8, No. 05; 2023

ISSN: 2456-3676

Material	PP
Grade Name	Lupol HI-4352L
Producer	LG Chemical
Comment	35%mineral ,D=1.17 g/cm3
Moldex3D Bank Version	2022.4.3

Process condition	
Melt temperature (minimum)	210 oC
Melt temperature (normal)	225 oC
Melt temperature (maximum)	240 oC

Figure 7. Temperature of Weld line

Comment: Based on data sheet of plastic. Min melting point is 210ºC. Considering the temperature of the weld, we see that its temperature during the filling process is greater than 210ºC (min temperature of the plastic), so these welds will not affect the product.

The position of the weld is in the middle and the position near the holes of the product

Reason: Where the currents meet; The air has no place to escape; Remedies; Increase injection speed and injection pressure; Check the mold's exhaust system or add more air holes; Use materials with low viscosity; Increase the injection port diameter.

5.2. Air trap

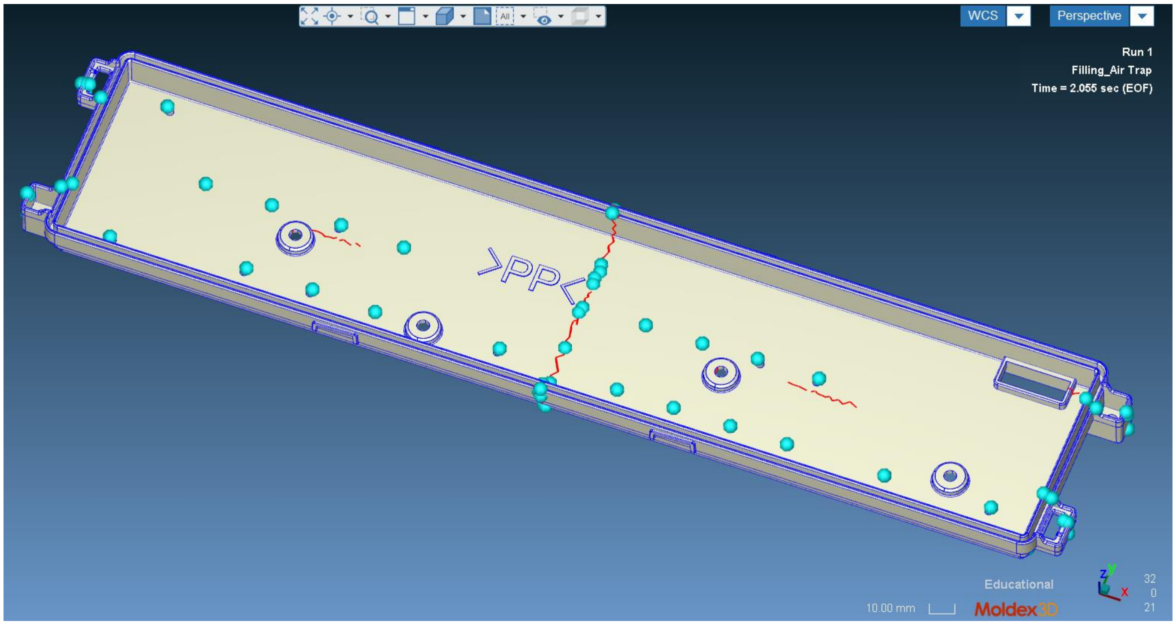


Figure 8. Air traps

Air traps are relatively abundant in the product, but they are mostly concentrated in the edges, ribs and edges of the pillars of the product; very little on the surface of the product. Therefore, we can fix it by making the inserts vent.

Reason: Raw materials have high moisture content but are not dried before being pressed; The position of the spray ports creates a concentrated flow leading to the accumulation of gas in one place; The product structure has many deep boss pillars that cannot be released; The channel air is not optimized, and the air in the mold cavity is not optimized, leading to air blockage.

Remedies: Make sure the plastic has been dried before being put into production; Change the injection port position; Design products with more suitable thickness; Optimizing the exhaust system in the mold; Reduce spray speed, because if sprayed at high speed, air bubbles cannot escape.

Conclusion: First of all, when making the mold, we will make the inserts to escape the air at the position of the cylinders. And the air outlet at the edge of the product so that the air at the edge can escape well.

6. Plastic channel

6.1. Calculating plastic channel according to theory

- Select cold channel type, Pin Point gate type for 3-plate mold.

• Spray peduncle, Silver spray peduncle

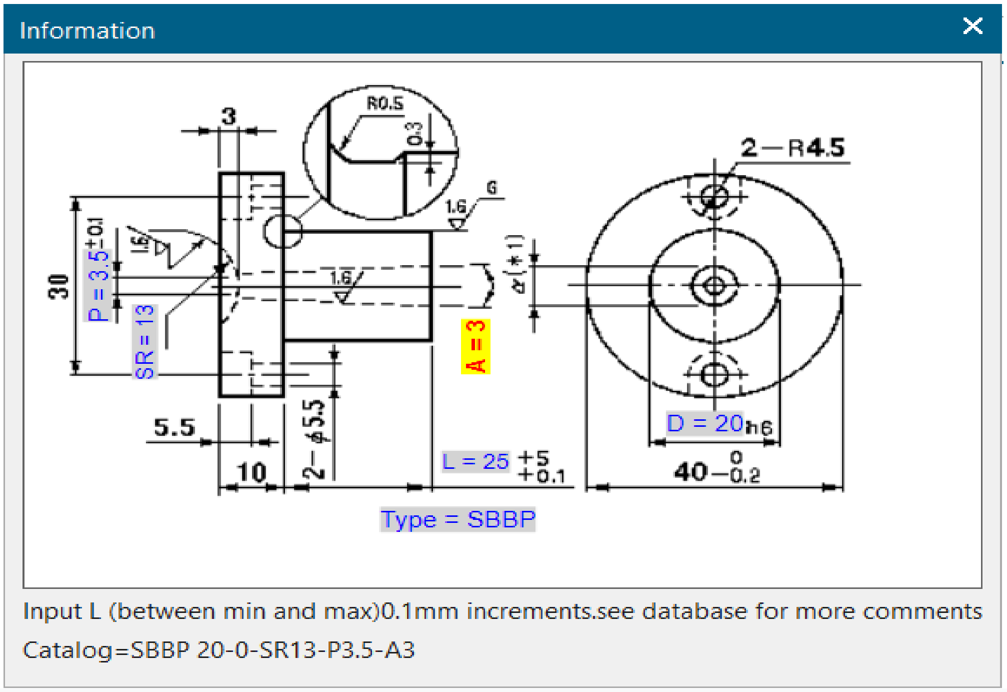


Figure 9. Silver stalk sprayed Misumi

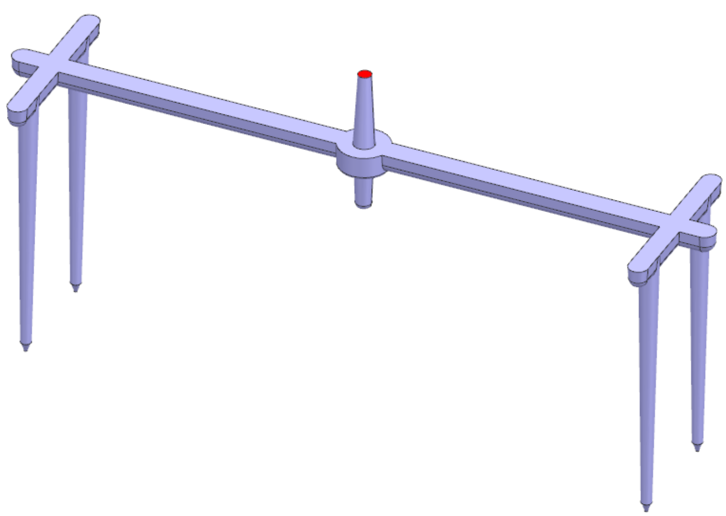


Figure 10. Runner according to the original design

Vol. 8, No. 05; 2023

ISSN: 2456-3676

6.2. Simulation results

There are two cases:

Case 1: Analysis according to the original design spray port and the original design water line.

Case 2: Analysis according to the optimal injection port indicated by the software. And increase the number of cooling channels.

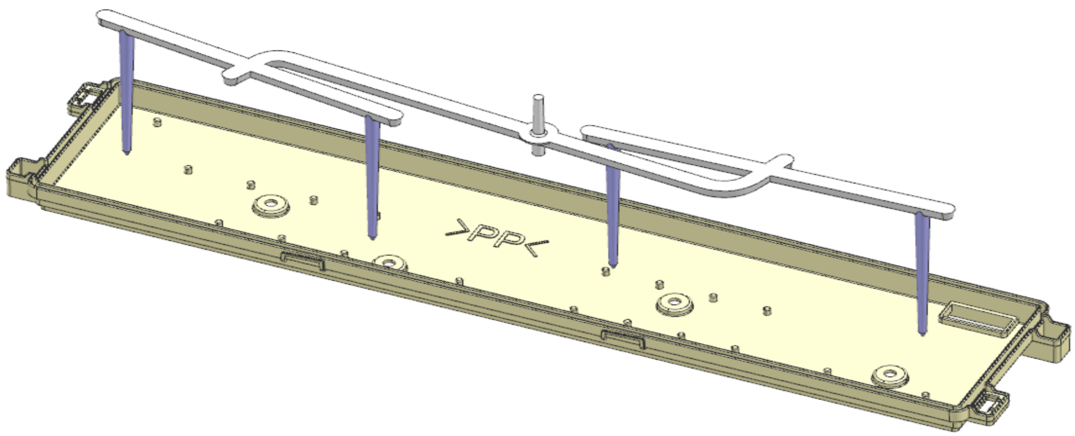


Figure 11. Runner by design software Modex3D for optimal analysis

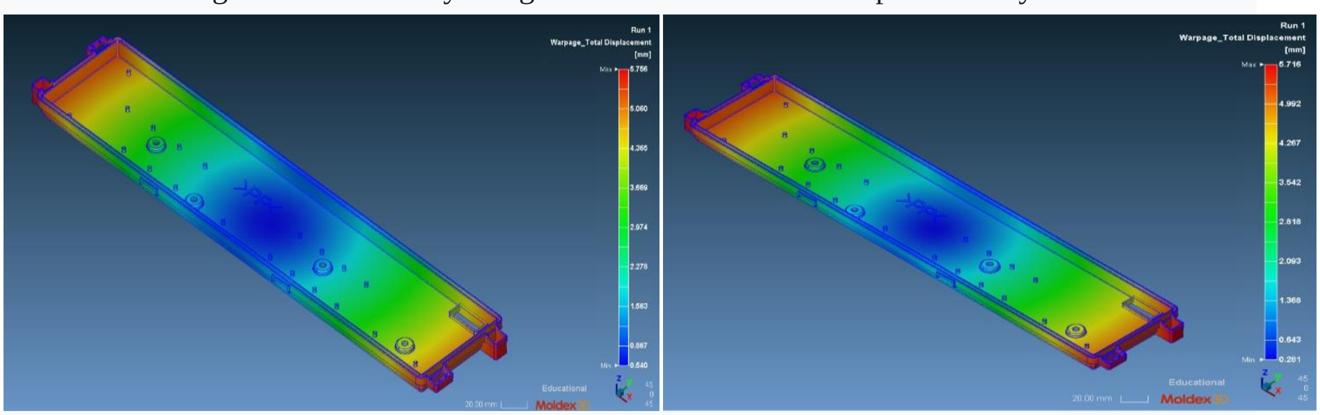


Figure 12. Fill time in original design (left) and in Modex3D (right)

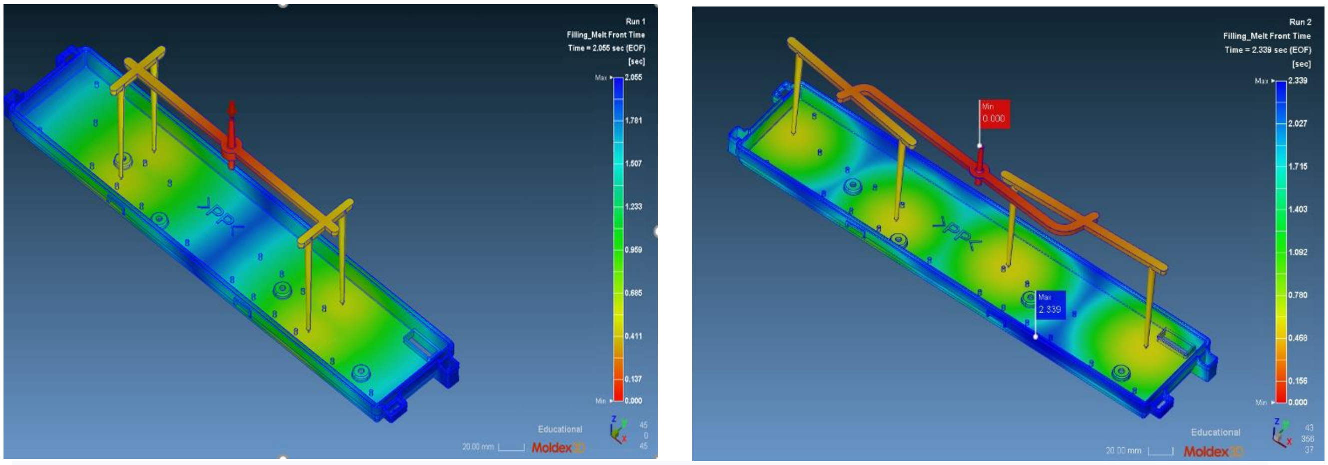


Figure 13. Warping when using original design (left) and according to Moldex3D (right)

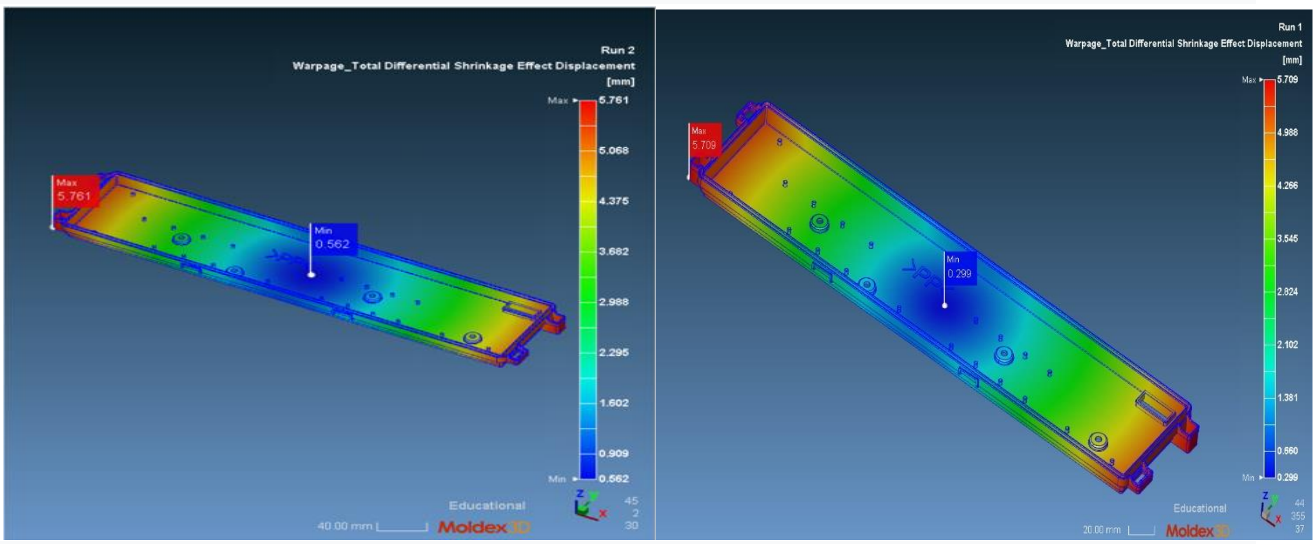


Figure 14. Warping due to pressure when using original design (left) and according to Moldex (right)

- The product is completely filled in both cases.

- When changing the injection port position, we see that the filling time increases from 2,055s to 2,238s.

- Product warping reduced from 5,756 mm to 5,716 mm

- Warping due to pressure of the product reduced from 5,761 mm to 5,709 mm

Conclusion:

- When the channel is arranged as shown by the Moldex3D software, the filling time is longer because the runner is longer than the injection pressure will be reduced. Warping due to pressure

Vol. 8, No. 05; 2023

ISSN: 2456-3676

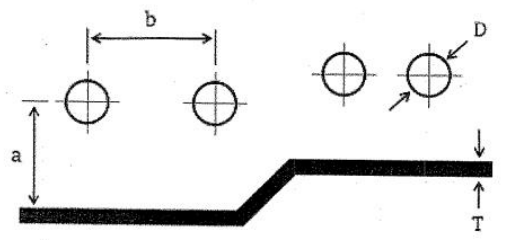
is reduced by 0.052 mm, which proves that when the arrangement is balanced between the spray ports, the product will be better protected.

- When arranging the injection port according to Moldex software, it is recommended that the warping of the product be reduced.

So we choose the design according to the Moldex software shown.

7. Cooling system

7.1 . Theoretical cooling system size calculation



T	D	a	b
~2 mm	8 ~ 10	2 ~ 3d	2 ~ 5d
2 ~ 4	10 ~ 12		
4 ~ 6	12 ~ 14		

Figure 15. Cooling channel size lookup table for design

Based on the figure below we have:

- Product thickness: W = 1.7 mm choose about 2mm.
- The diameter of the cooling channel is taken as 10 mm.
- Distance from the cooling channel plate to the product wall: a = 2 x 10 = 20 mm.
- Distance between two cold channel plates: b = 4.5 x 10 = 45 mm

7.2. Design of cooling channel system

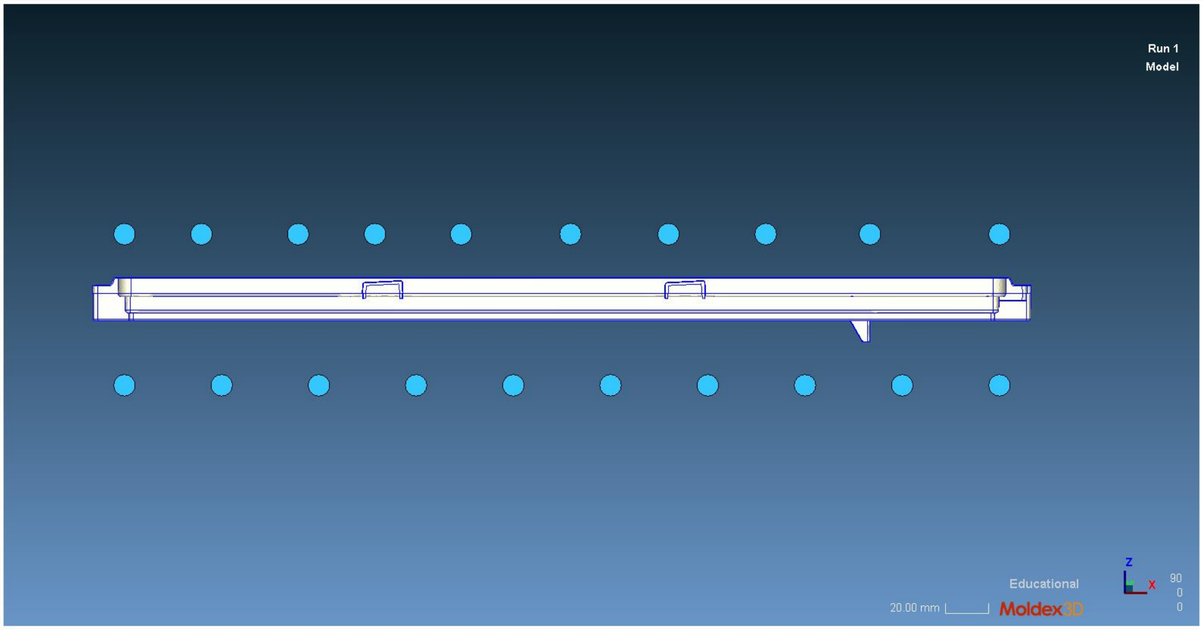
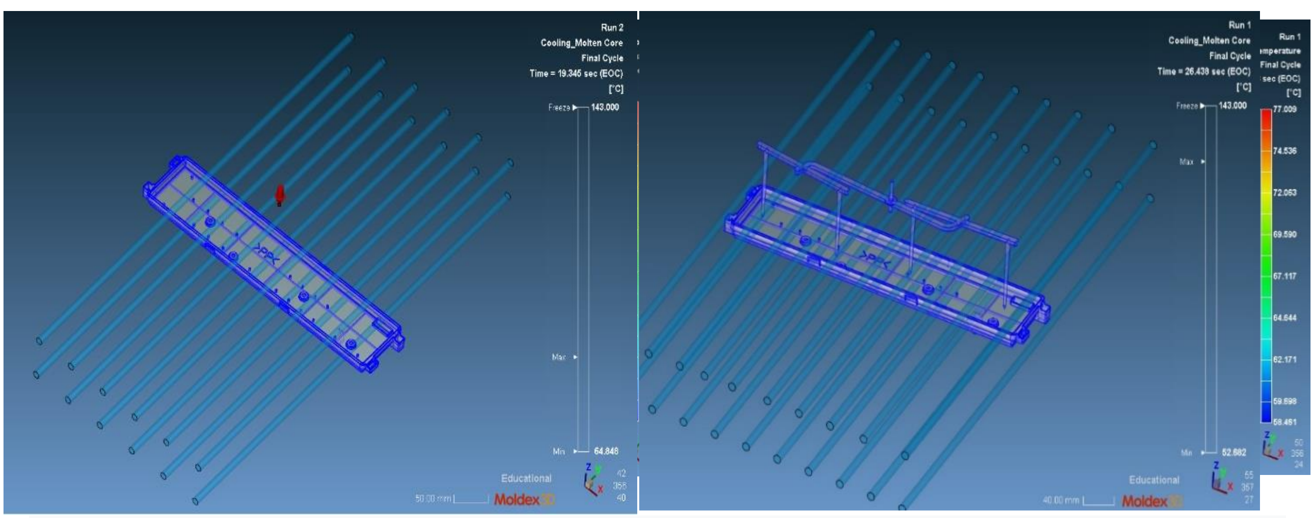


Figure 16. Optimum cooling channel system. Arrange 10 channels above, below

7.3. Simulation results

- Case 1: Arrange a small number of cooling channels. 8 channels above and below.



- Case 2: Arrange a small number of cooling channels. 10 channels above and below.

Figure 17. Cooling temperature of cooling channel by case 1 (left) and by field case (right side

Figure 18. Molten Core of cooling channel by case 1 (left) and by case 2 (right)

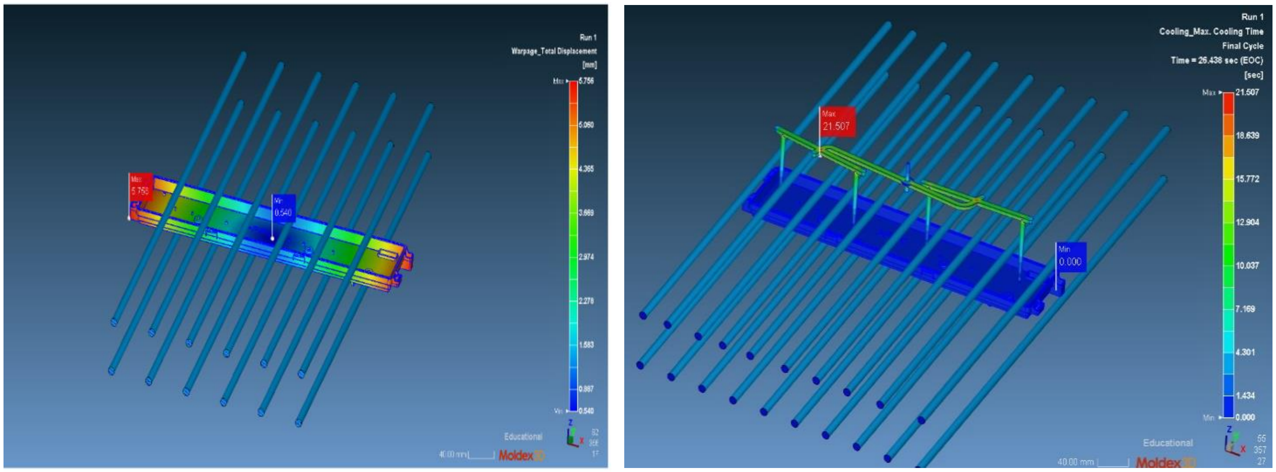


Figure 19. Maximum cooling time of cooling channel by case 1 (left) and case 2 (right)

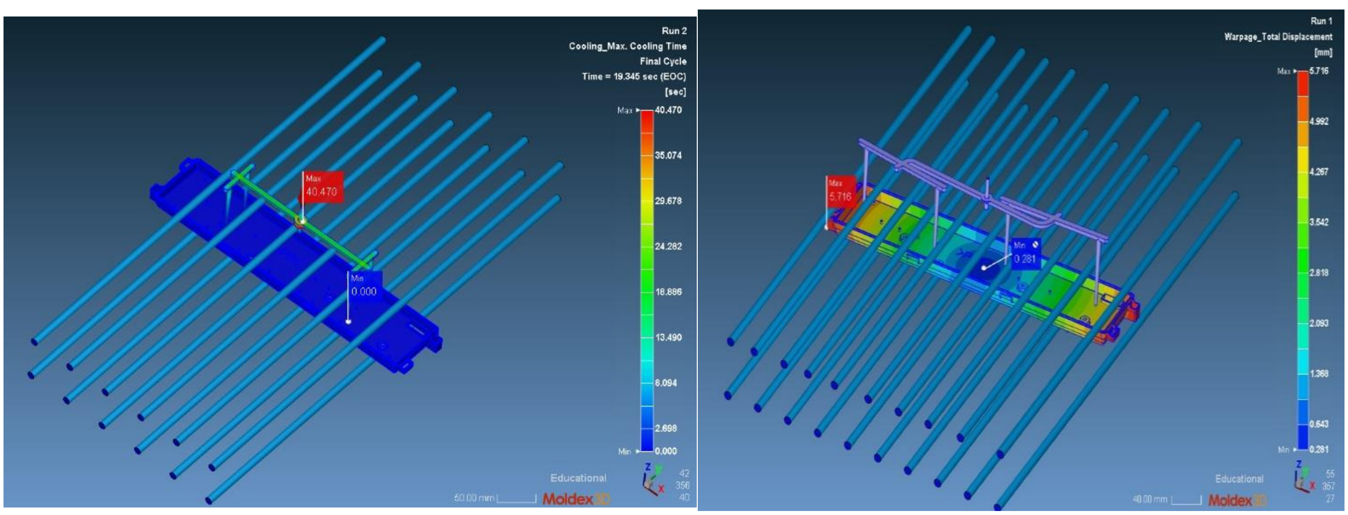


Figure 20. Warping when using cooling channel according to case 1 (left) and case 2 (right)

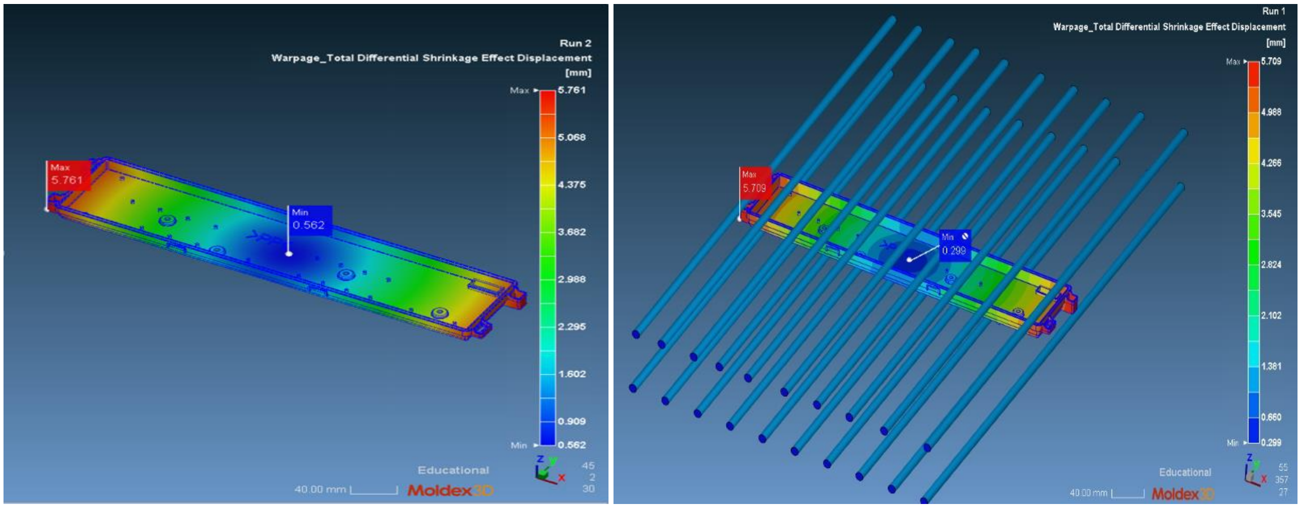


Figure 21. Warping due to overpressure when using cooling channel according to case 1 (left) and case 2 (right)

- The product is completely filled and solidified in both cases.
- By design case 2 has more cooling channels than case 1
- Cooling temperature according to case 1 design is not as good as case cooling 2
- Case 2 cooling time is faster than case 1
- Warping of the product according to the design of case 2 is smaller than that of case 1, reduced from 5,756 mm to 5,716 mm
- Warping due to pressure of the product in case 2 is smaller than in case 1, reduced from 5,761 mm to 5,709 mm.

The temperature difference of the cooling channel according to the design of case 1 (85,469°C) is larger than the difference of the temperature of the cooling channel according to the case 2 (77,009°C)

The cooling channel system by design using more cooling channels reduces cooling time, and reduces total warping. In addition, the smaller temperature difference between the cooling channels should indicate a better cooling system. So we choose the cooling system according to case 2.

Vol. 8, No. 05; 2023

ISSN: 2456-3676

8. Product Improvement

8.1 Change gate position.

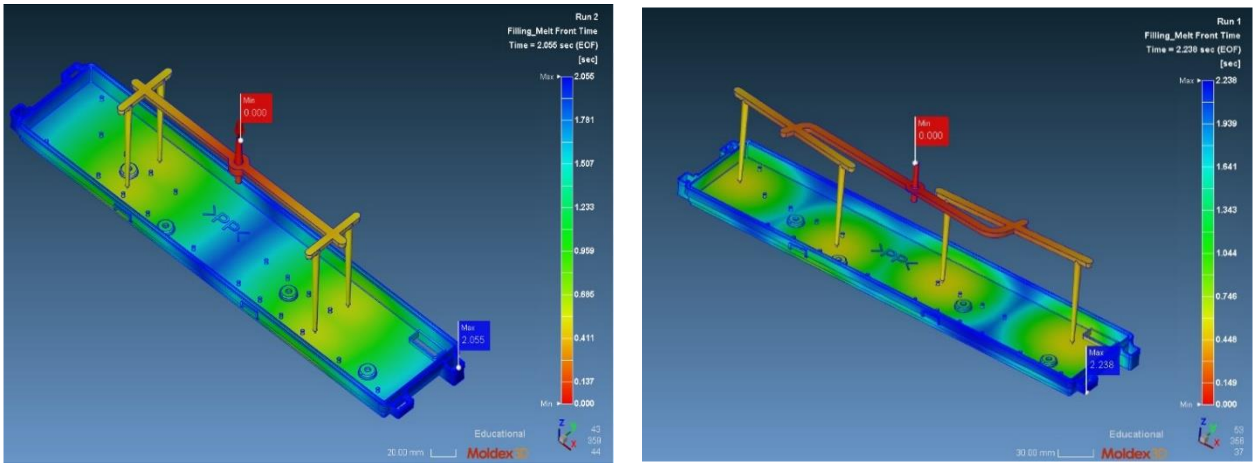


Figure 22. Filling time according to the original gate design (left) and according to Moldex3D (right)

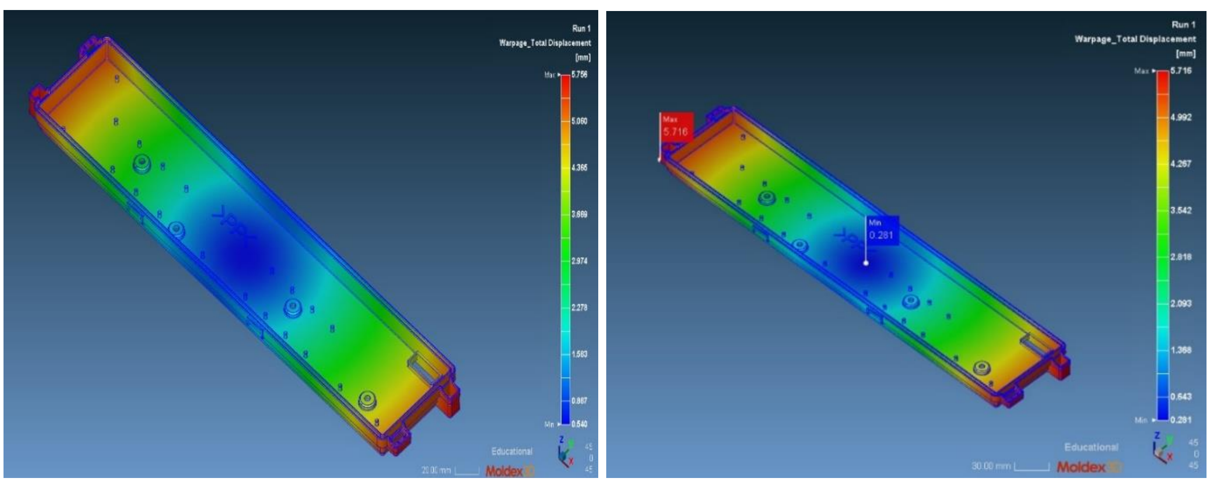


Figure 23. Warping according to the original gate design (left) and according to Moldex3D (right)

- When changing the gate according to Moldex3D software, we see that the warping is reduced. From 5,756 mm down to 5,716 mm

- Therefore, we will choose to design according to Moldex software.

ISSN: 2456-3676

8.2. Increased pressure protection from 70 to 90%

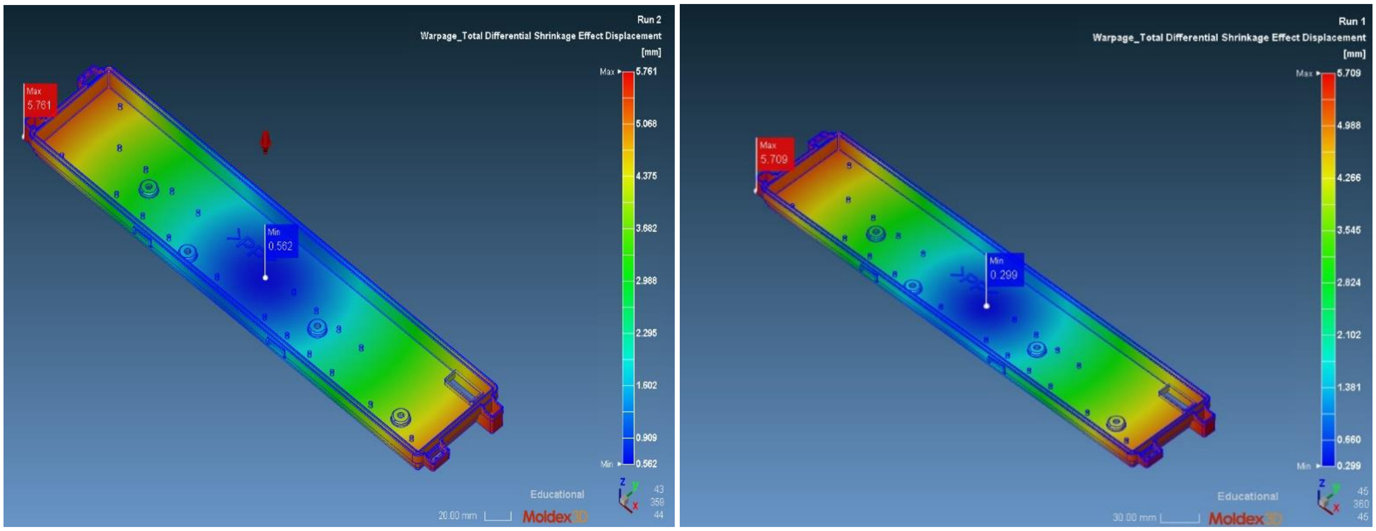


Figure 24. Warping due to pressurization when the pressure is 70% (left) and 90% (right)

When the pressure is increased to 20%, the warping on the product is significantly reduced (5,761 mm to 5,709 mm), this warping has not yet met the technical requirements. Still need to continue to improve.

8.3. Increase mold temperature (from 40°C to 80°C)

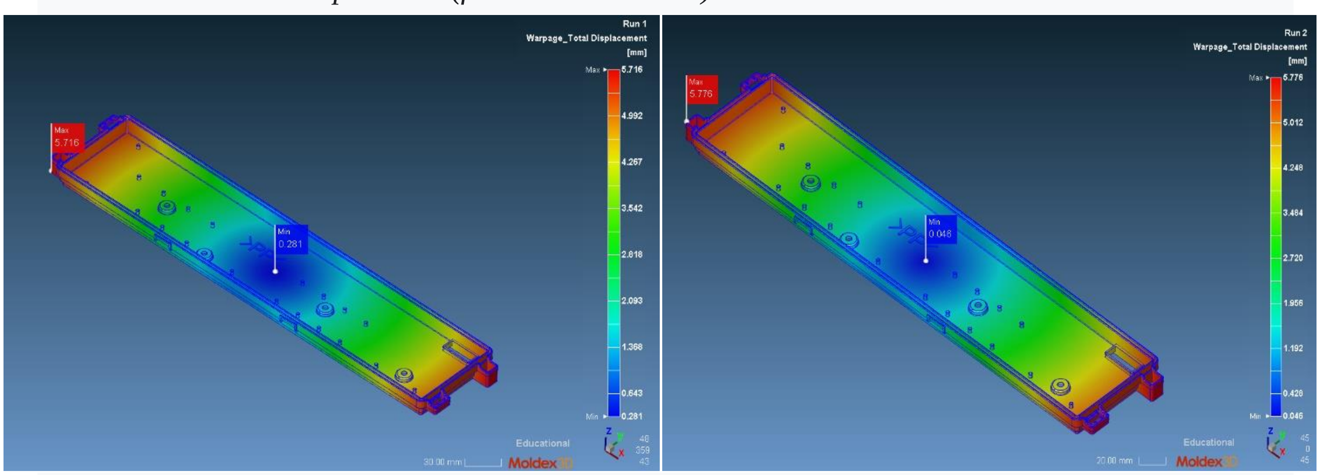


Figure 25. Warping when mold temperature is 40°C (left) and 80°C (right)

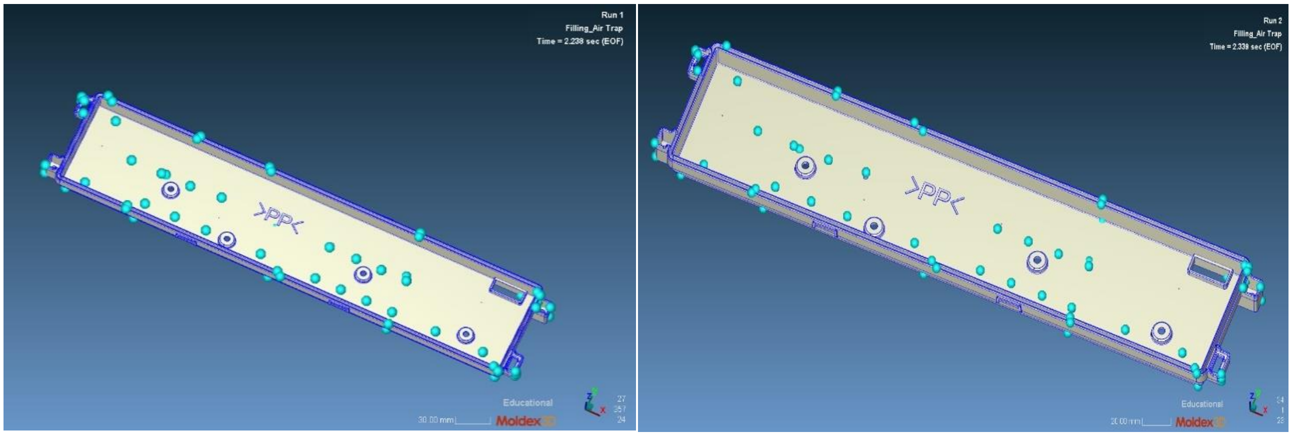


Figure 26. Air bubbles when mold temperature is 40oC (left) and 80oC (right)

When increasing the mold temperature from 40°C- to 80°C, we see greater warping, and more air bubbles appear. As the mold temperature increases, the shrinkage increases. This leads to increased warping, and a faster drop in plastic viscosity, resulting in air escaping in time, resulting in more air bubbles.

So we still keep the mold temperature at 40 °C because it can be seen that increasing the mold temperature does not improve the warping of the product but also increases the amount of air bubbles.

8.4. Conclusion.

After improving the gate position, cooling water line, pressure protection, mold temperature, we see that the product has a change in warping, but it is not significant.

After analysis and exchange, work with the customer again. To reduce product warping, it is necessary to change the product design such as increasing product thickness, adding stiffening ribs to the product.

Summary

To analyze the CAE for a plastic product, that is, the part obtained before the injection molding process to create a plastic product with the help of a software tool, it is necessary to follow the following process: Analyze and describe the requirements. Technical requirements of plastic products pressed by plastic injection molds are about to be completed; CAE analysis to find the best injection port location; Design and select the specifications of the most suitable plastic channel on the plastic injection mold; Optimum exhaust system to reduce surface burn or air-popping errors on plastic products; And offer a plan to improve the product or improve the structure of the plastic injection mold to satisfy the product's criteria and reduce errors such as welding errors, warping errors, air trap errors...

And the results of the above work help the designer get the most optimal design for the plastic injection mold while minimizing the testing costs and reducing the mold testing time before making the plastic injection mold.

References

Chan WM, Yan L, Xiang W, Cheok BT (2003) A *3D CAD knowledge-based assisted injection mould design system*. Int J Adv Manuf Technol 22:387–395.

Deng YM, Britton GA, Lam YC, Ma YS (2002) *Feature-based CAD/CAE integration model for injection-moulded product design*. Int J Prod Res 15:3737–3750.

Godec D, Sercer M, Osrecki (2009) *Design of mould for injection moulding of moulded part with internal thread*, Tech Gazzete 16(1):53–62.

Hodolic J, Matin I, Stevic M, Vukelic DJ (2009) *Development of integrated CAD/CAE system of mold design for plastic injection molding*, Mater Plastice 46(3):236–242.

Huang JM, Jou YT, Zhang LC, Wang ST, Huang CX (2009), *A web-based model for developing: a mold base design system*, Expert Syst Appl 36(4):8356–8367.

Jong WR, Wu CH, Liu HH, Li MY (2009), *A collaborative navigation system for concurente mold design*, Int J Adv Manuf Technol 40(3–4):215–225.

Kong L, Fuh JYH, Lee KS, Liu XL, Ling LS, Zhang YF, Nee AYC (2003), *A Windows-native 3D plastic injection mold design system*, J Mater Process Technol 139:81–89.

Low MLH, Lee KS (2003), *Application of standardization for initial design of plastic injection moulds*, Int J Prod Res 41:2301–2324.

Maican E, Bayer M, Tabara A, Balasoiu V (2008), *Optimization of a polymeric rod from plants conveying equipment*, Mater Plastice 45(2):184–196.

Nardin B, Kuzman K, Kampus Z (2002), *Injection moulding simulation results as an input to the injection moulding process*, Int J of Mater Process Technol 130–131:310–314.

Zhou H, Shi S, Ma B (2009), *A virtual injection molding system based on numerical simulation*, Int J Manuf Technol 40:297–306.