ISSN: 2456-3676

Improve Product Design and Injection Molding for Electrical Outlet Cover Plastic with the Help of CAE Software

Nguyen Thi Le Hang[1], Nguyen Duc Chinh[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.2023.8612 URL: http://dx.doi.org/10.51505/ijaemr.2023.8612

Received: Nov 29, 2023 Accepted: Dec 05, 2023 Online Published: Dec 13, 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.

Vol. 8, No. 06; 2023

ISSN: 2456-3676

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 analysisbefore 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: Uniform product thickness. Product warping ≤ 0.2mm. Make sure to mix it with the Frame details. After installation, there is no warping, the gaps are small and uniform.
- Appearance requirements: Product thickness is beautiful, no scratches, no plastic defects (weld lines, burn marks, incomplete filling...), spray port marks on the bottom of the product.

2. Product material, mass and volume

- PC materials
- Density: 1.2 g/cc
- Lotte chemical company
- Plastic name: Infino CF-1050
- Product volume: 7620,760 mm³
- Product weight: 9.145 grams

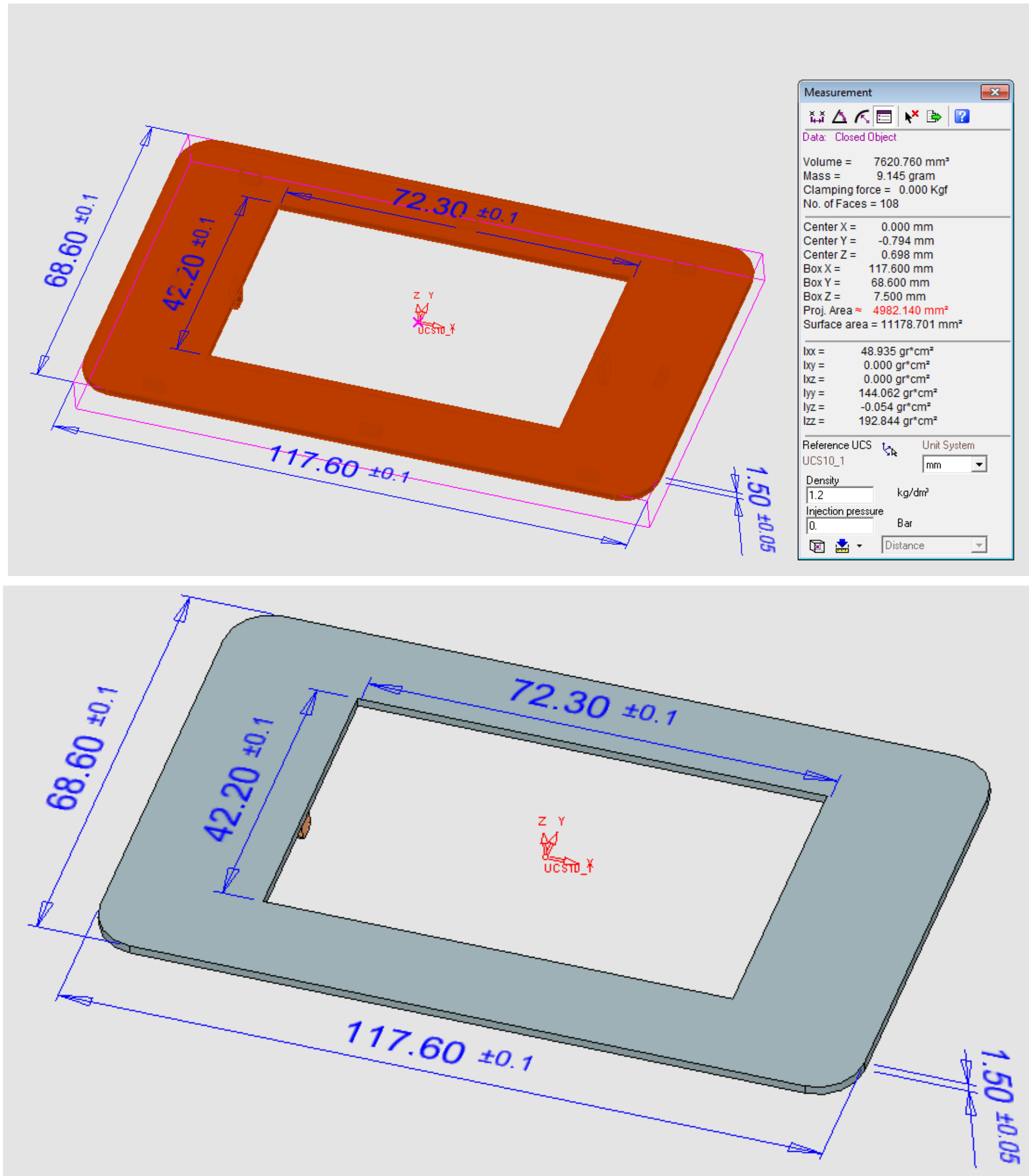


Figure 1. 3D of Product

3. Product thickness

Check product thickness based on design in NX software.

Vol. 8, No. 06; 2023

ISSN: 2456-3676

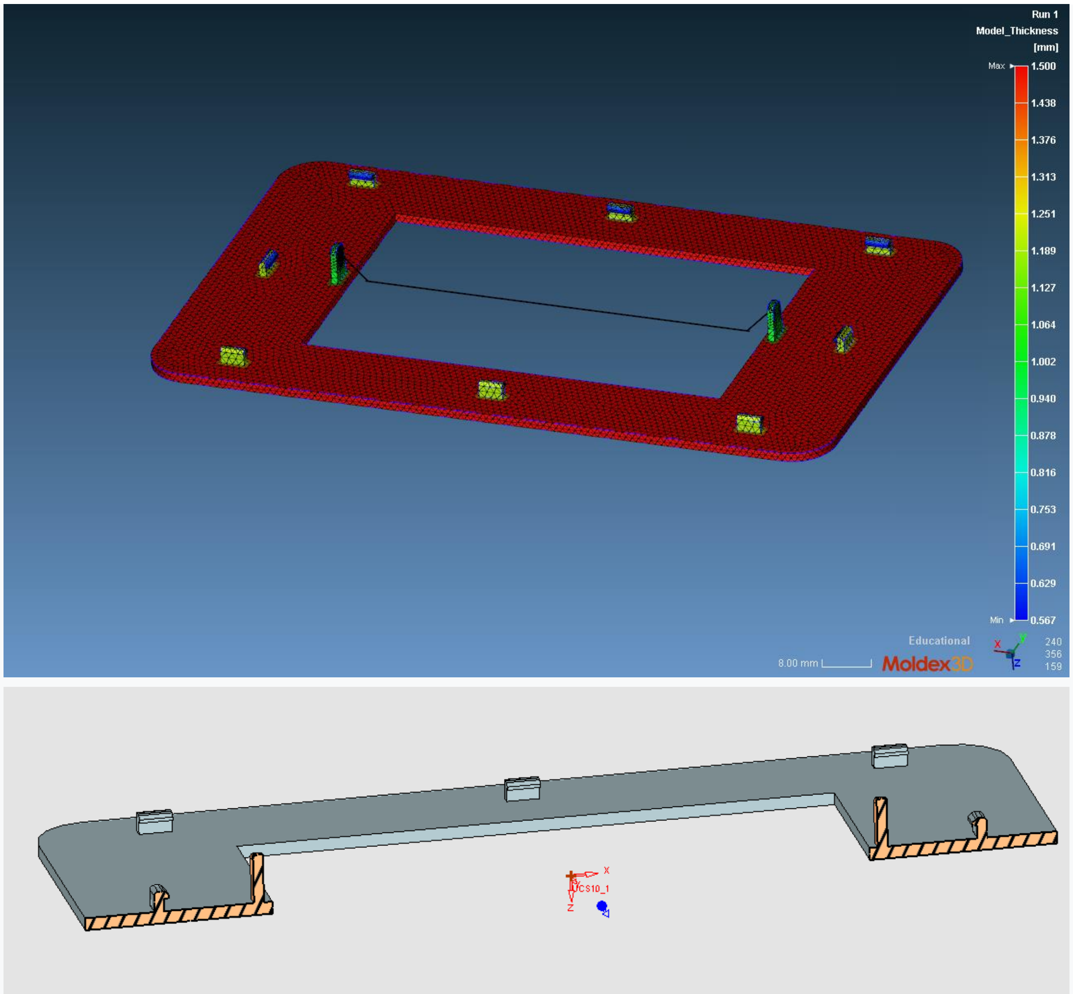


Figure 2. Check product thickness with software

Maximum product thickness is 1.5mm; Minimum product thickness is 0.567 mm.

4. CAE finds the best injection port location

Analyzing the best plastic injection port from Moldex3D 2022 software we see.

- 1 injection port option: long filling time, easy to generate air bubbles, weld line on the opposite side of the pump port. So 1 injection port will not be optimal.

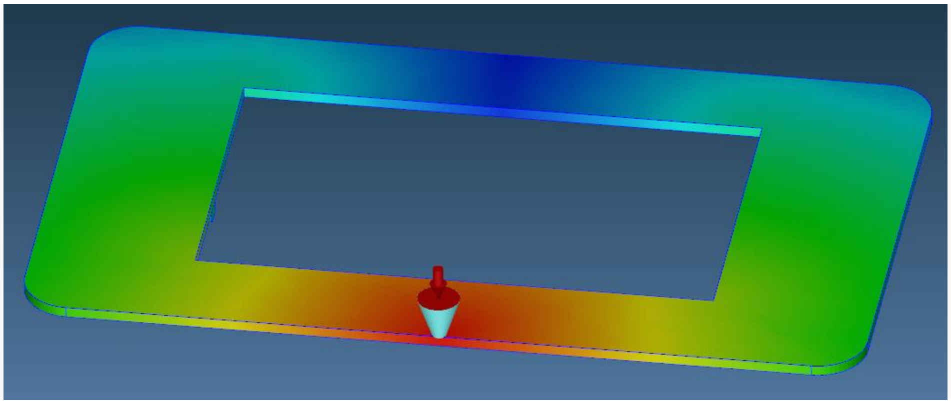


Figure 3. One injection port.

Two injection ports option: In the analysis position of Moldex3D 2022 software, we have 3 ways to place the pump port.

+ Method 1: Place 2 positions as the quilt suggests. Because the mold structure is a 2-plate mold, cold runner, and the location of the injection port is not exposed on the outer surface of the product. So this method is not chosen.

+ Method 2: Place next to the product. Because the plastic is ABS, spraying gate size easily generates jets at the injection gate position. So method 2 is not reasonable.

+ Method 3: Tunnel gate: This position sprays on the back of the product, limiting spraying errors.

=> Among 3 ways to choose spray location. We find method 3 is the most optimal.

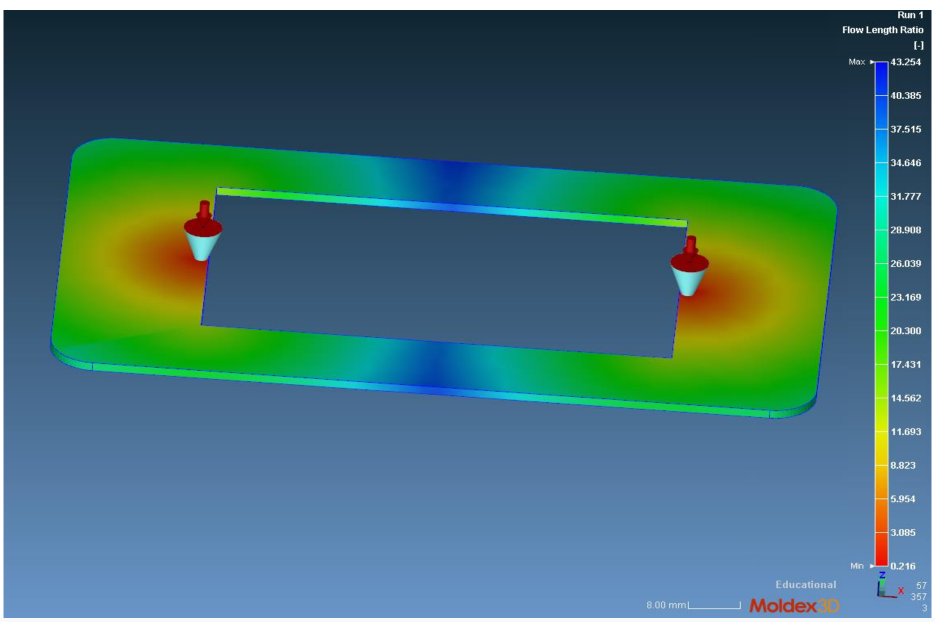


Figure 4. Spray port location analyzed by Moldex3D 2022 software.

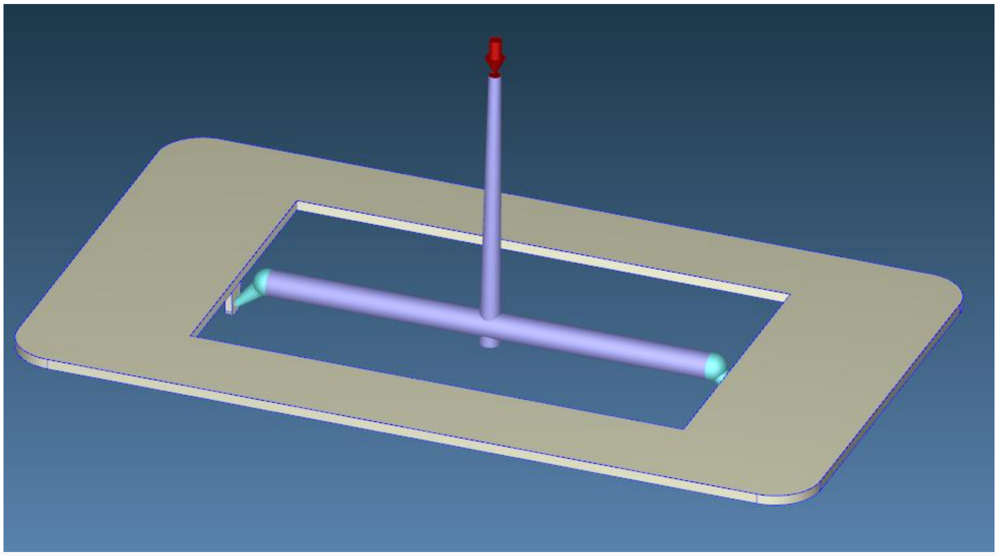


Figure 5. The most optimal location of the injection port

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

5.1. Welding line

Welding simulation results.

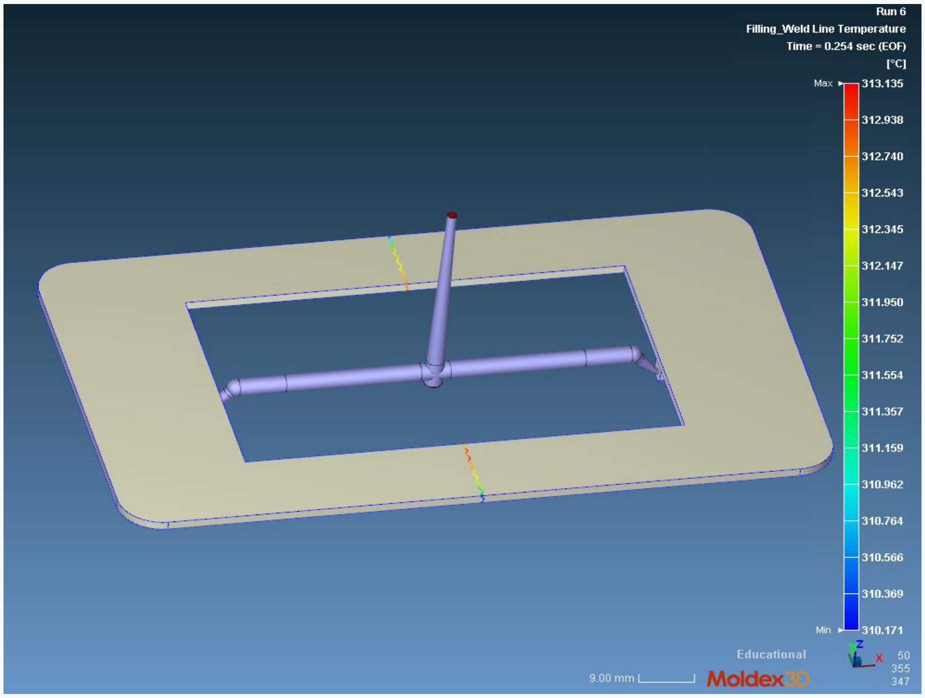


Figure 6. In front of product

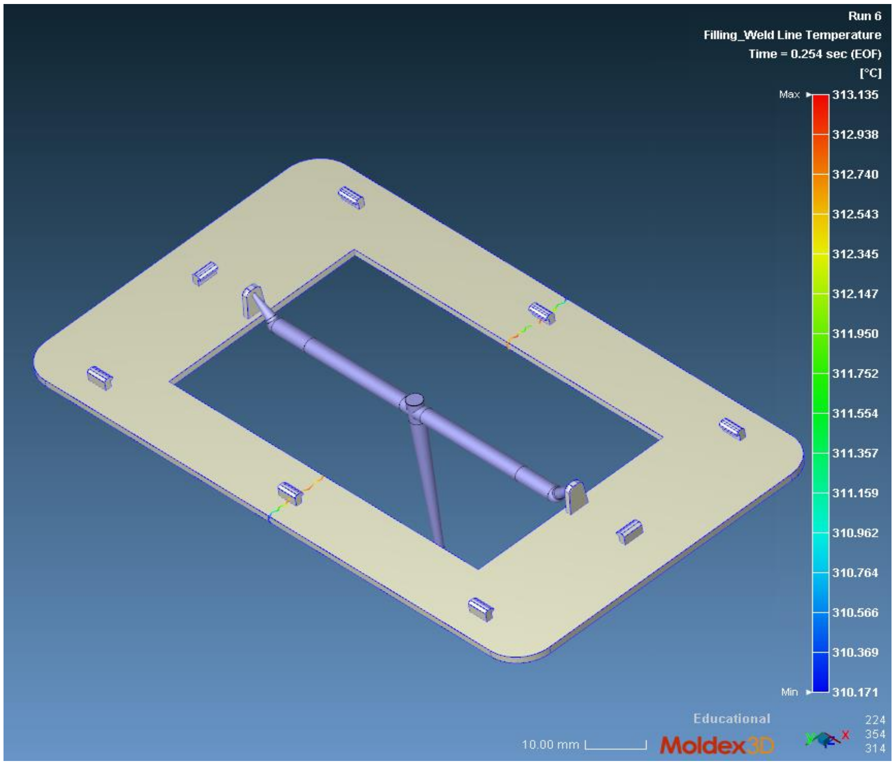


Figure 7. Behind of product

The welding position is in the middle of the product. Plastic temperature range at the weld seam is 310.171ºC-313.135ºC. PC plastic melting temperature is 260ºC-310 ºC so the weld line will not be obvious.

5.2. Air trap

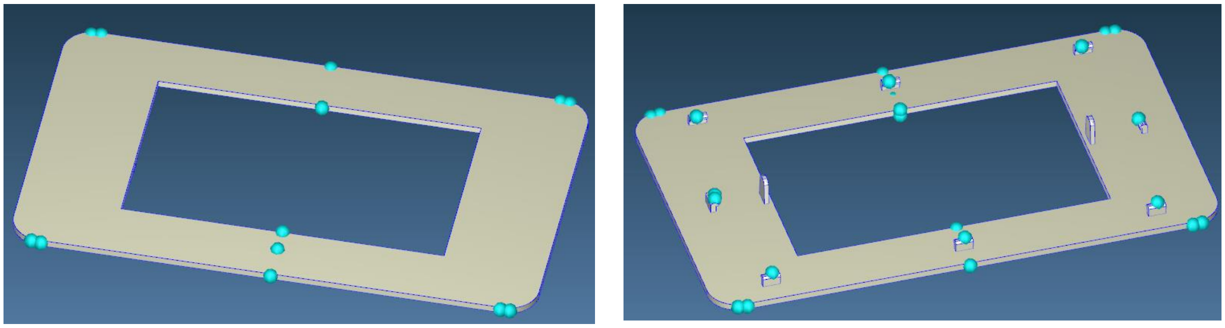


Figure 8. Air traps

Comment: Air bubbles are relatively abundant in the product, but they are mostly concentrated located on the edges, ribs and edges of the product; very little on the product surface. Therefore, we can fix it with very little damage to the product surface.

- Reason:

+ Raw materials have high moisture content but must not be dried before pressing.

+ The position of the injection ports creates a concentrated flow, leading to air being concentrated in one place.

+ The spray nozzle, plastic channel, and plastic entrance gate have low gloss, are rough and are rough. It is easy for air bubbles to form when molten plastic passes through.

+ The channel air release and air release in the mold cavity have not been optimized, leading to air blockage.

- Remedies:

+ Make sure the plastic has been dried before being put into production.

+ Optimize the air release system in the mold.

+ Reduce the spray speed, because if sprayed at high speed, air bubbles cannot escape.

+ The surface of the channel, spray nozzle and plastic pouring gate have a gloss level that meets the given level allowed, no burrs or rough surfaces.

6. Plastic channel

6.1. Calculating plastic channel according to theory

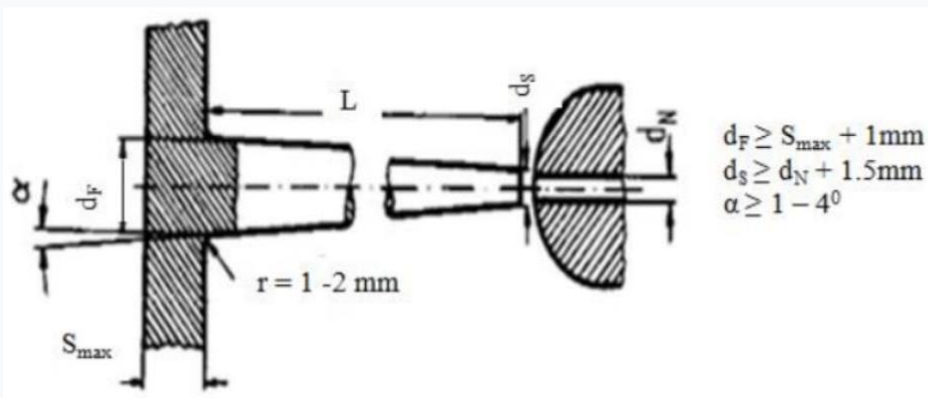


Figure 9. Spray stem size for design.

JSW hiroshima Plant J140AD-110H-A → dN= 2 mm.

$dF \geq Smax + 1\ mm \geq 2 + 1 = 3mm$; $dF = 4\ mm$.

$ds \geq dN + (1\text{ to }1.5)\ mm \geq 2 + (1\text{ to }1.5) = (3\text{ to }3.5)\ mm$; $ds = 4.5\ mm$.

$\alpha \geq 1 - 4°$; $\alpha = 1°$.

L = 45 mm.

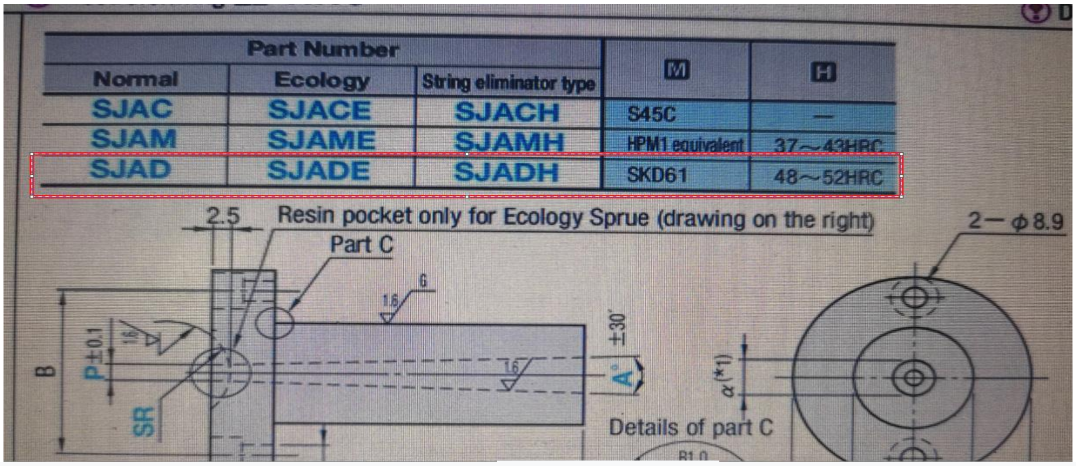
Silver spray stem:

ISSN: 2456-3676

JSW hiroshima Plant J140AD-110H-A → $dN = 2$ mm
$dF \geq Smax + 1mm \geq 2.5 + 1 = 3.5\ mm$; $dF = 4\ mm$
$ds \geq dN +$ (1 to 1.5)$mm \geq 2 +$ (1 to 1.5) = (3 to 3.5) mm; $ds = 4\ mm$.

$\alpha \geq 1 - 4°$; $\alpha = 1°$**;** L = 45 mm.

The Misumi catalog to choose it:



Part Number			M	H
Normal	Ecology	String eliminator type		
SJAC	SJACE	SJACH	S45C	—
SJAM	SJAME	SJAMH	HPM1 equivalent	37~43HRC
SJAD	SJADE	SJADH	SKD61	48~52HRC

Figure 10. Silver spray stem

6.2. Simulation results

Product filling time is 0.254s.

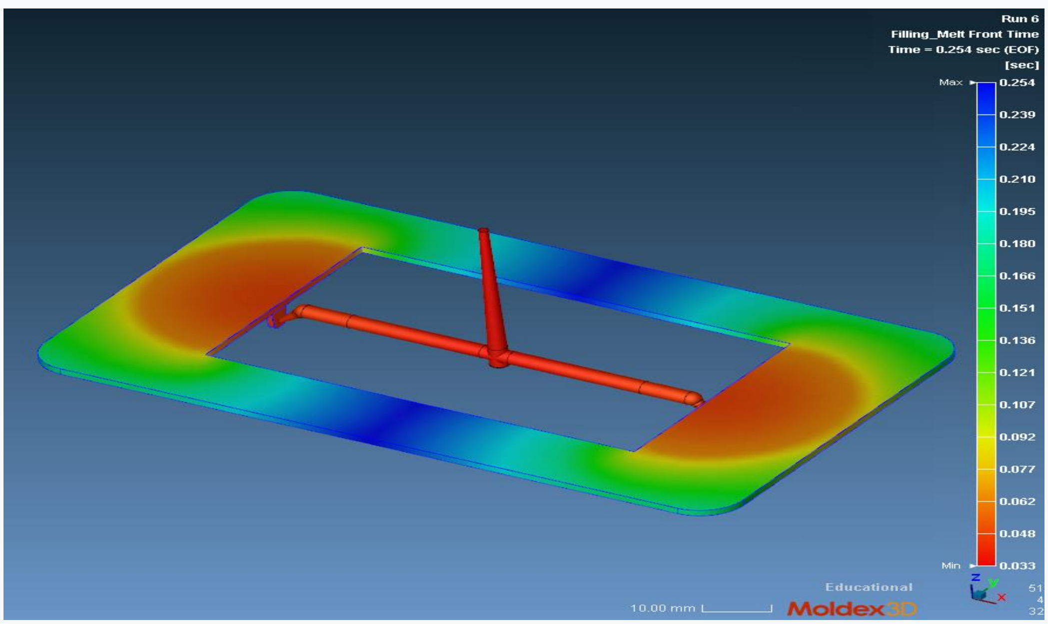


Figure 11. Filling time at injection port diameter is 0.8mm.

- The maximum product warping is 0.293mm

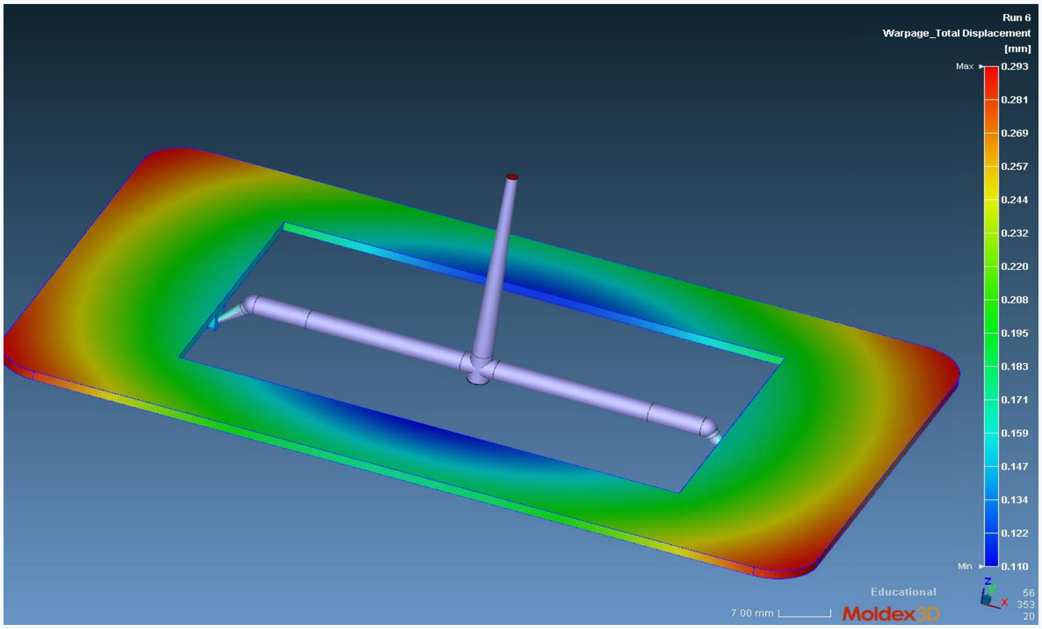


Figure 12. Warping when using injection port diameter of 1.2mm.

+ Warping due to pressure 0.258mm:

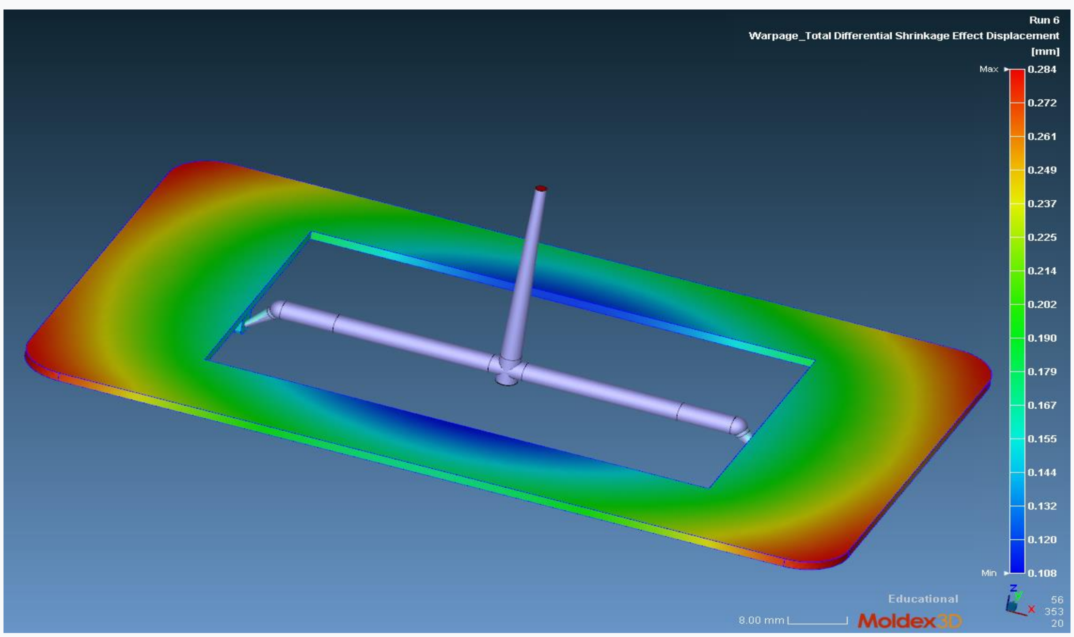


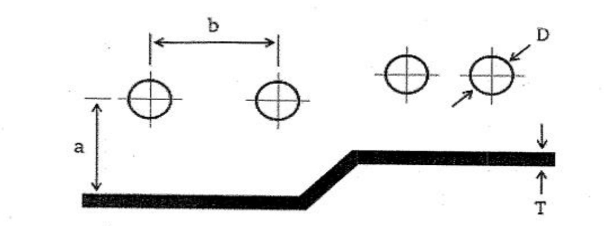
Figure 13. Warping due to pressure

Vol. 8, No. 06; 2023

ISSN: 2456-3676

7. Cooling system

7.1. Theoretical cooling system size calculation



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

Figure 14. Cooling channel size lookup table for design

Based on the figure below we have:

- Product thickness: W = 1.5 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 8 = 16 mm.
- Distance between two cold channel plates: b >= 3.75 x 8 = 30 mm.

7.2. Design of cooling channel system

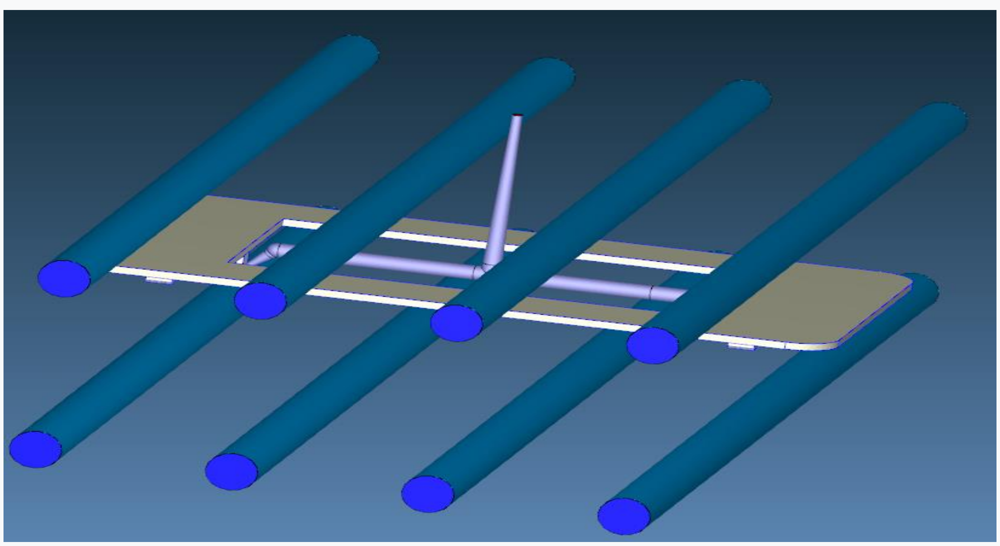


Figure 15. Optimum cooling channel system.

ISSN: 2456-3676

7.3. Simulation results

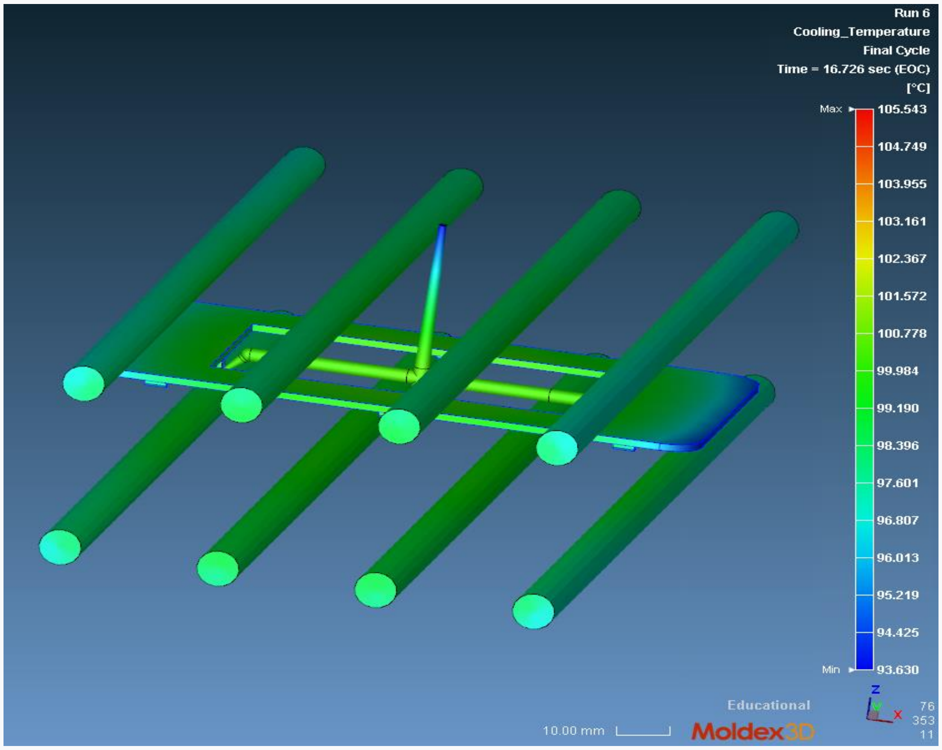


Figure 16. Cooling temperature of cooling channel

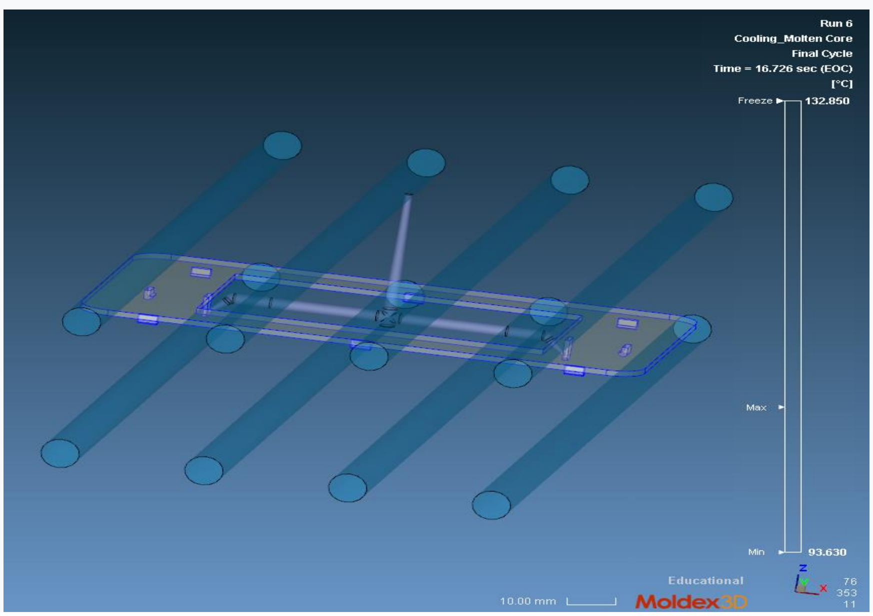


Figure 17. Molten Core of cooling channel by design.

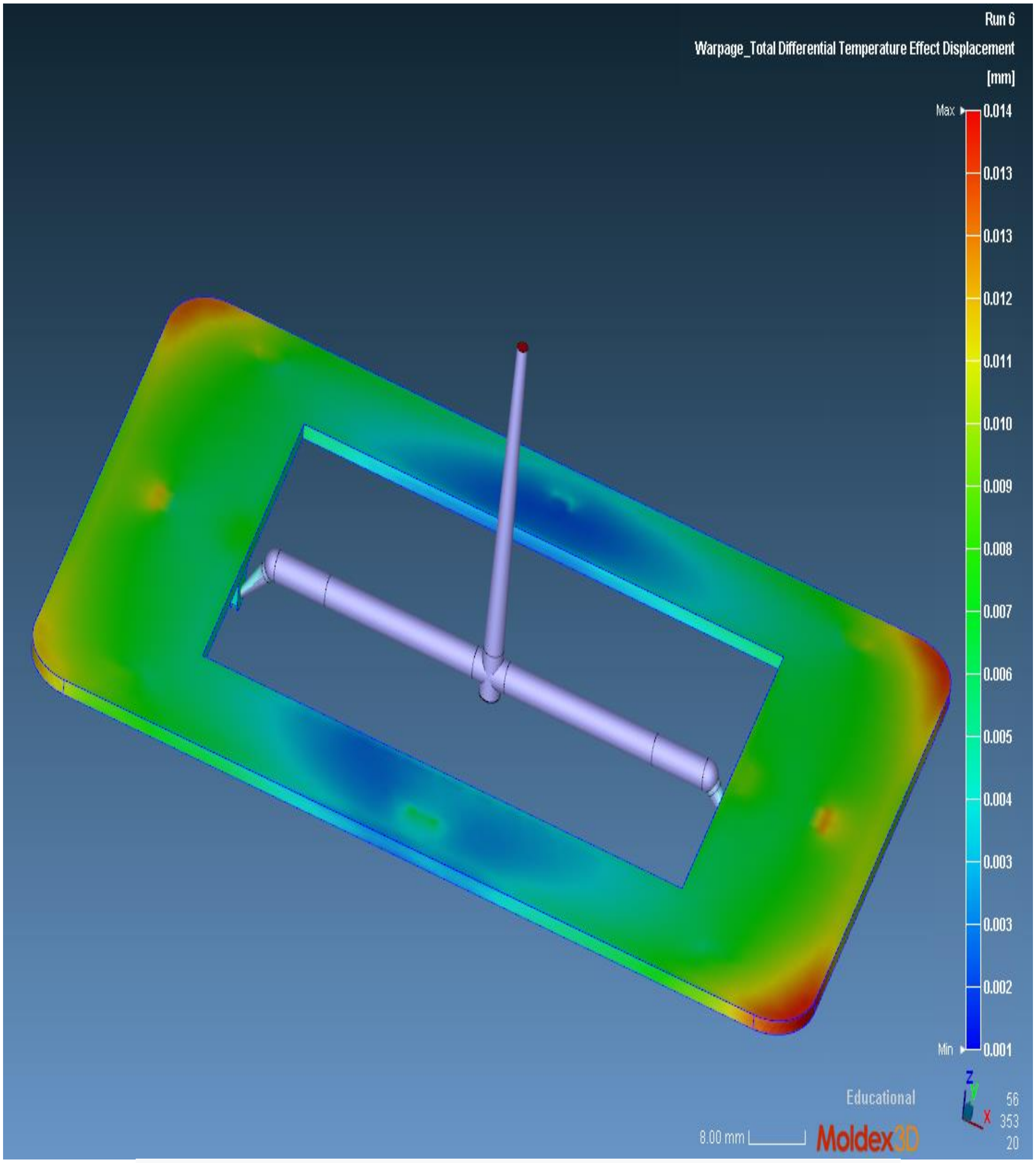


Figure 18. Warping when using cooling channel according to design

Comments:

- The product is completely cooled and solidified.
- Warping due to cooling 0.014mm.
- Cooling meets requirements.

8. Product Improvement

8.1. Concave product surface

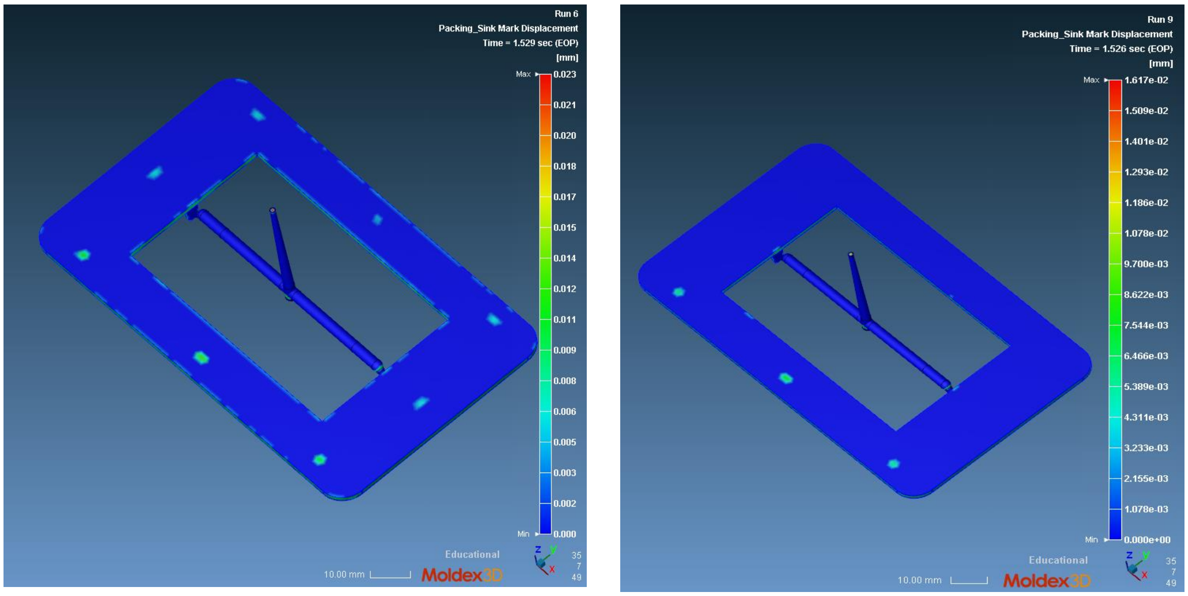


Figure 19. Increase compression pressure from 75% (left) to 90% (right).

Result:

- Concave product surface at 75% pressure max 0.023mm.
- Concave product surface at 90% pressure max 0.016mm.

Conclusion: Maximum surface concavity of 0.016mm is invisible to the naked eye. Appearance meets requirements.

8.2. Product warping.

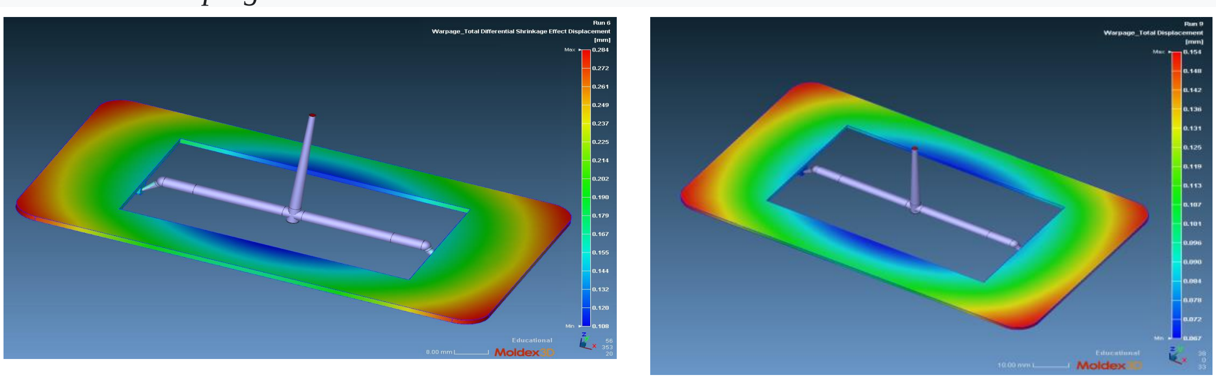


Figure 20. Warping due to pressurization when the pressure is 75% (left) and 90% (right)

Result:

- Product warping at 75% pressure max 0.284mm.
- Product warping at 90% pressure max 0.154mm.

Conclusion: Warping meets product standards ≤ 0.2mm.

8.3. 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.