

DESIGN AND MANUFACTURE OF INJECTION MOLDS FOR WALL-MOUNTED PLASTIC HOOK PRODUCTS

Tran Duy Nam^{1*}

¹*Dong Nai Technology University.*

*Corresponding author: *Tran Duy Nam, tranduynam@dntu.edu.vn.*

GENERAL INFORMATION

Received date: 10/01/2025

Revised date: 14/04/2025

Accepted date: 03/07/2025

KEYWORD

Design;

Simulation;

Fabrication.

ABSTRACT

This project focuses on the research, design, and fabrication of a two-plate injection mold with a slider mechanism to produce wall-mounted plastic hook products. Our research and design approach adheres to standards and fabrication practices to create a complete mold with dimensions of 320 mm × 250 mm × 240 mm, using S50C steel as the mold material. The dimensions of Product 1 are 72.5 mm × 23 mm × 38 mm, Product 2 are 37 mm × 35 mm × 6.5 mm, and Product 3 are 29.5 mm × 22 mm × 7.5 mm, all made from PP plastic. We utilized Inventor 2024, Creo Parametric 8.0, and Moldex 3D software for design and simulation. The achieved outcome is the successful creation of a two-plate mold model with the mechanisms and functions as proposed.

1. INTRODUCTION

Nowadays, the plastics industry has become one of the fastest-growing sectors, driven by the increasing consumer demand for plastic products. Plastic products and components are widely used across various fields such as healthcare, education, electronics, telecommunications, transportation, agriculture, and fisheries. (Ni'am et al., 2017; Mahajan et al., 2018; Masato et al., 2018) This trend has significantly fostered the robust development of the mold design and manufacturing industry.

In line with the global growth of the plastics industry, domestic production and processing of plastic materials have also gained attention and development. A notable achievement is the emergence of a wide variety of plastic products, with continuous improvements in quality and

complexity. Additionally (Park et al., 2010; Montalvo et al., 2015; Uyen et al., 2020; Carlson et al., 1996; Yao et al., 2002) machinery and equipment systems have been progressively upgraded and enhanced.

However, during their studies and work,(Chen et al., 2003; Park et al., 2006; Nian et al., 2015; Song et al., 2007) the research team realized that merely approaching and studying injection molds for plastic products through theoretical and simulation-based methods is insufficient. Many students still face challenges in understanding the structure and operational principles of certain types of molds. Therefore, integrating mold models into direct teaching is essential to improve the quality of education and inspire students' passion for mold design and manufacturing.

With the aim of applying learned knowledge and practical experience gained during internships, the team strives to design and manufacture a complete mold to solidify and enhance their knowledge while creating a foundation for innovative ideas to promote the development of the mold-making industry.

For these reasons, the research team decided to propose and implement the study, design, and manufacturing of an injection mold for the “wall-mounted plastic hook” product.

Abbreviations and Symbols

- PP: Polypropylene
- PE: Polyethylene
- ABS: Acrylonitrile Butadiene Styrene
- POM: Polyoxymethylene
- PA: Polyamide
- PPS: Polyphenylene Sulfide
- PC: Polycarbonate
- PET: Polyethylene Terephthalate
- PS: Polystyrene
- CAE: Computer-Aided Engineering
- EMX: Expert Moldbase Extension
- CAD: Computer-Aided Design
- CAM: Computer-Aided Manufacturing
- CNC: Computer Numerical Control

Material and Process Parameters

- $T_{\text{solidification}}$: Solidification temperature of the material ($^{\circ}\text{C}$)
- T_{ejection} : Ejection temperature of the product from the mold ($^{\circ}\text{C}$)
- CTE: Coefficient of thermal expansion of plastic material ($\text{cm}/^{\circ}\text{C}$)

Forces

- F_{friction} : Friction force
- F_{normal} : Normal reaction force (N)
- F_{eject} : Ejection force (N)

Geometrical and Mechanical Properties

- Draft angle: Draft angle Φ
- A_{eff} : Effective cross-sectional area of the part (mm^2)
- E: Elastic modulus

Machining Parameters

- n: Spindle speed (revolutions/minute)
- V_c : Cutting speed (m/minute)
- D: Milling cutter diameter

2. METHODOLOGY

Our research and design approach adheres to standards and fabrication practices to create a complete mold with dimensions of $320 \text{ mm} \times 250 \text{ mm} \times 240 \text{ mm}$, using S50C steel as the mold material.

3. FINDINGS AND DISCUSSION

3.1. Theory

3.1.1. Polypropylene (PP) Material

The material selected by the group for this project is PP (Polypropylene), a commonly used thermoplastic. PP possesses high mechanical strength, is typically white, odorless, tasteless, and non-toxic.

Thanks to its advantages of being easy to synthesize, non-toxic, and cost-effective, PP is widely utilized across various industries in all countries and for diverse products.

PP has extensive applications in numerous fields. Some common uses of PP include:

- Packaging

- Healthcare
- Electrical and Electronics Engineering
- Automotive parts manufacturing
- Additionally, PP is also widely applied in the production of household items and furniture.

3.1.2. General Structure of an Injection Mold

A mold comprises multiple components assembled together, typically divided into two main parts:

Core (Movable Mold Part): This part is mounted on the movable plate of the injection molding machine.

Cavity (Fixed Mold Part): This part is mounted on the fixed plate of the injection molding machine.

The mold cavity determines the external shape of the product, while the core determines the internal shape. A mold may have one or multiple cavities and cores depending on the design requirements. The interface where the cavity and core meet is called the **parting surface**.

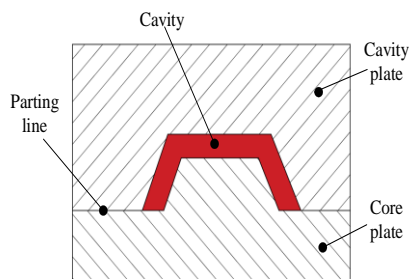


Figure 1. Core and Cavity in the Closed State

In addition to the core and cavity, the mold consists of several other components that are assembled to form the basic mold system, including:

Guiding and Positioning System: Comprising guide pins, guide bushes, locating rings, positioning units, return pins, etc., this system ensures the accurate alignment of the two

mold parts when assembled to create a precise cavity.

Plastic Flow System: Includes the sprue bushing, runners, and gate, responsible for supplying molten plastic from the injection nozzle into the mold cavity.

Ejection System: Comprising ejector pins, return pins, support pins, ejector sleeves, ejector plates, retaining plates, and support blocks, this system functions to eject the product from the mold after the injection process is completed.

Side Core System: Consists of side cores, core slides, cam guide bars, inclined pins, hydraulic cylinders, etc., designed to release undercuts in the mold along the mold opening direction.

Ventilation System: Includes vent grooves that expel trapped air from the mold cavity, facilitating the smooth flow of molten plastic into the cavity and preventing defects like air bubbles or burning marks on the product.

Cooling System: Includes water channels, grooves, heat-conducting pipes, and connectors, tasked with stabilizing the mold temperature and cooling the product efficiently.

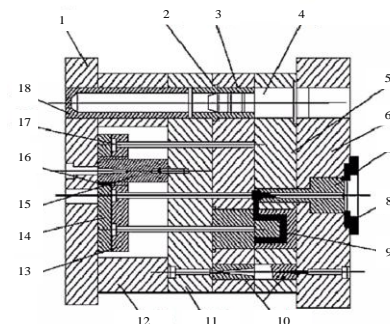


Figure 2. General Structure of a Mold

1. Rear Clamping Plate
2. Guide Bush
3. Core Plate (Movable Mold Plate)
4. Guide Pin

5. Cavity Plate (Fixed Mold Plate)
6. Front Clamping Plate
7. Sprue Bushing
8. Locating Ring
9. Product
10. Positioning Unit
11. Support Plate
12. Support Block
13. Retaining Plate
14. Ejector Plate
15. Support Pin
16. Guide Bushing
17. Return Pin
18. Spacer Sleeve

3.2. Product design

Creo Parametric 8.0 is a software developed by Parametric Technology Corporation, founded in Boston, Massachusetts, in 1985. This software is highly robust for designing and manufacturing injection molds, offering comprehensive modules from detailed part design to mold assembly and integrated mold cavity machining programming. These features simplify the mold design process significantly.

Creo's intuitive interface, simple operations, fast processing speed, and capability for simulating plastic flow make it an effective tool for injection mold design today.

Design Specifications for the “Wall-Mount Plastic Hook”

- Dimensions (L × W × H): 72.5 mm × 23 mm × 38 mm
- Maximum Thickness: 5 mm
- Material: Polypropylene (PP)



Figure 3. Plastic Hook Detail

Design Specifications for the “Hook Clip”

- Dimensions (L × W × H): 37.5 mm × 35.5 mm × 6.5 mm
- Maximum Thickness: 1.5 mm
- Material: Polypropylene (PP)

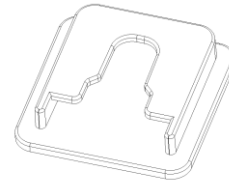


Figure 4. Hook Clip

Design Specifications for the “Hook Clip”

- Dimensions (L × W × H): 30 mm × 22 mm × 7.5 mm
- Maximum Thickness: 5 mm
- Material: Polypropylene (PP)

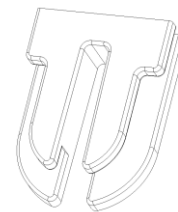


Figure 5. Clasp

* Mold design

Ejector System

After the product in the mold cools down, the mold is opened. At this point, the product is still stuck to the mold cavity due to vacuum suction, and the product tends to shrink after cooling. Therefore, an ejector system is required to remove the product.

For the group's design, a pin ejector system is used. To distribute the force evenly over the surface, the ejector pins are evenly spaced along the product. Here, 3-mm diameter ejector pins with a length of 125 mm, according to Misumi standards, are used. The ejection force is calculated by the formula in:

$$F_{eject} = \cos(\Phi) \cdot \mu_s \cdot F_{normal}$$

Calculation of Ejection Force for the "Wall Hook" Part

$$F_{eject} = \cos(\Phi) \cdot \mu_s \cdot E \cdot CTE \cdot (T_{solidification} - F_{ejection}) \cdot A_{eff}$$

The number of ejector pins required: 5, pin diameter: 3 mm The stress on the ejector pin applied to the product: 4.89 Mpa

The calculated stress is less than the tensile strength of PP plastic, so the ejector force will not damage the product.

The calculated stress is less than the tensile strength of PP plastic, so the ejector force will not damage the product.

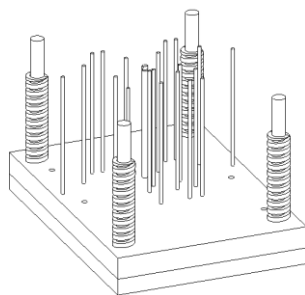


Figure 6. Ejector System

Slider System

In this design, the group uses an undercut opening system with an inclined pin according to Misumi standards [5].

The inclined pin is selected with an inclination angle of $\alpha = 24^\circ$ and a diameter of $d = 12$ mm.

Based on Figure 7, the length of the inclined pin is calculated:

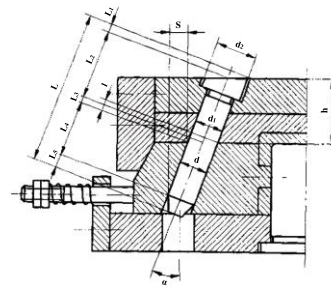


Figure 7. Length of the Inclined Pin

$$L = L_1 + L_2 + L_3 + L_4 + L_5 \tag{3}$$

$$= \frac{d_2}{2 \tan \alpha} + \frac{h}{\cos \alpha} + \frac{d}{2} \tan \alpha + \frac{s}{\sin \alpha} + (5 \div 10) = \frac{12}{2 \tan 24} + \frac{15}{\cos 24} + \frac{12}{2} \tan 24 + \frac{40,68}{\sin 24} + (5 \div 10) = 127 \text{ mm}$$

The slider is selected with the angle of the locking block being larger than the angle of the inclined pin, so the mold can fully close and avoid collisions during the molding process. The inclination angle of the locking block is $\beta = 26^\circ$.

• Sprue:

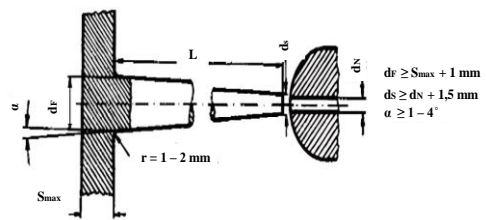


Figure 8. Sprue size

We have: $d_F \geq S_{max} + 1 \tag{4}$

$$\Leftrightarrow d_F \geq 3,5 + 1 = 4,5 \text{ mm}$$

$$D_s \geq d_N + 1,5 \tag{5}$$

$$\Leftrightarrow D_s \geq 2 + 1,5 = 3,5 \text{ mm}$$

$$\alpha = 1,5^\circ$$

Select the runner with a circular cross-section, and the runner diameter according to the theory::

$$t = 6 \Rightarrow D = 6 + 1,5 = 7,5 \text{ mm}$$

However, since the product has small dimensions, a short flow length, and the injection gate is located at the thickest part of the product for easy filling, the runner diameter is selected as 3.5 mm. CAE software will be used to recheck the product's fillability.

3.3. Simulation

Table 1. Properties of PP Plastic

Property	Value
Density	0,78 - 2,77 g/cc
Rockwell Hardness R	33 - 112
Tensile Strength (Maximum)	33 - 112
Tensile Strength (Maximum)	5 - 128 MPa
Tensile Strength at Yield	18 - 27 MPa
Water Absorption	0,01 - 0,2%
Elongation at Break	250 - 700%
Modulus of Elasticity	0,1 - 6,2 GPa
Flexural Strength	25 - 50 MPa
Resistivity	50 - 1e+ 17 ohm-cm
Surface Resistivity	80 - 1e+16ohm
Processing Temperature	87,8 - 280°C
Melting Temperature	145 - 302°C
Solidification Temperature	121°C

Moldex software is a leading CAE tool in the field of plastic flow simulation today. The software helps users minimize product defects,

predict common errors, and failures that often occur during the injection molding process. Based on the properties of PP plastic in Table 1, we can obtain the simulation results as shown below.

3.3.1. Filling step

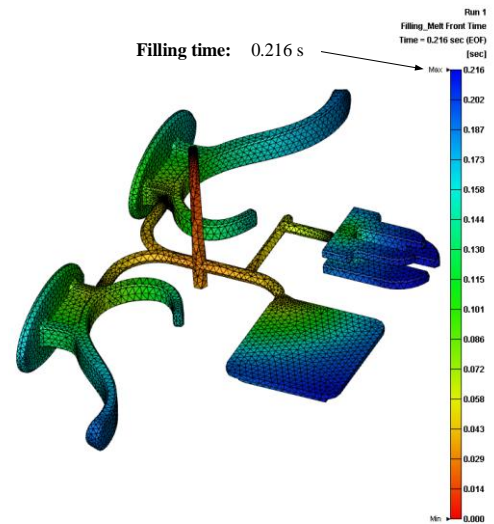


Figure 9. Filling step

Through the simulation results, we can see that the filling time of the product is quite short. Based on the time distribution graph, we observe that the filling time is distributed quite evenly, with the exception of the final point of the product, which is colored green and takes more time to fill than the other parts.

With a total filling time of 0.216 seconds, the filling result is reasonable. The plastic is distributed evenly across the entire product, with no signs of material shortage. The filling time is quite short.

3.3.2. Air trap

Based on the analysis results in Figure 10, we can see that the air pockets are mainly distributed at the ends of the part, and the gaps in the slider opening system. Therefore, with the mold design chosen by the team, most of the air pockets in the simulation can be avoided. In this

case, the air will be evacuated by the open slider structure..

requirements, this level of shrinkage is acceptable.

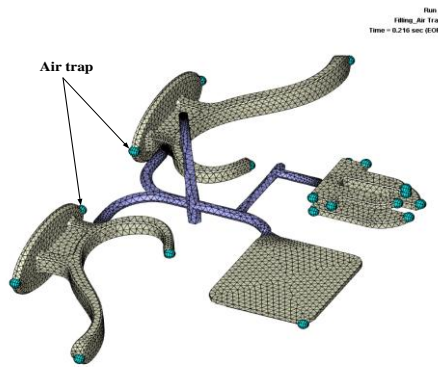


Figure 10. Air trap

3.3.3. Pressure

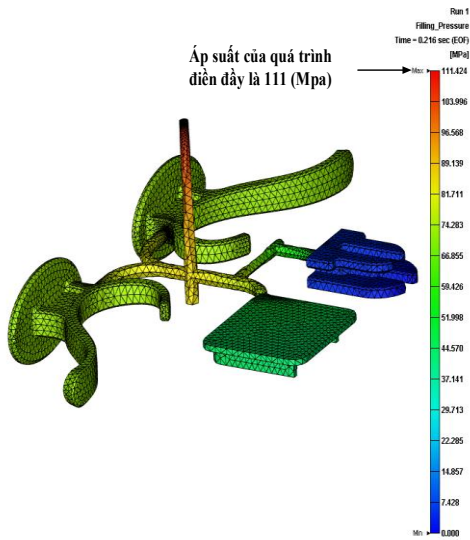


Figure 11. Injection pressure

From the simulation results in Figure 11, we can see that the filling pressure is 111.424 MPa. The required injection nozzle pressure is 105.793 MPa, which is suitable for many types of injection molding machines currently available.

3.3.4. Shrinkage

From the simulation in Figure 12, we can see that the shrinkage of the product ranges from -0.5% to 2.2%. Since the product is not a mating part and does not have high dimensional

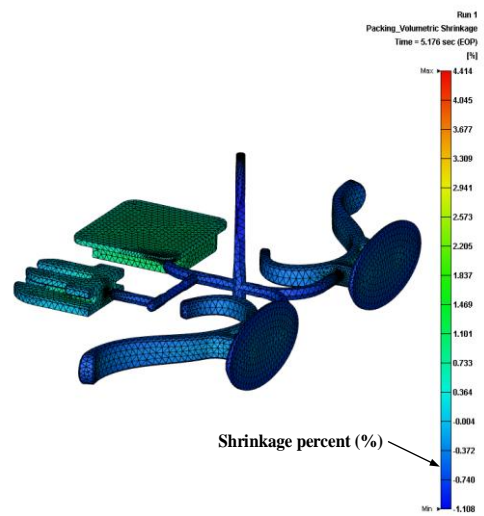


Figure 12. Part shrinkage

3.3.5. Cooling step

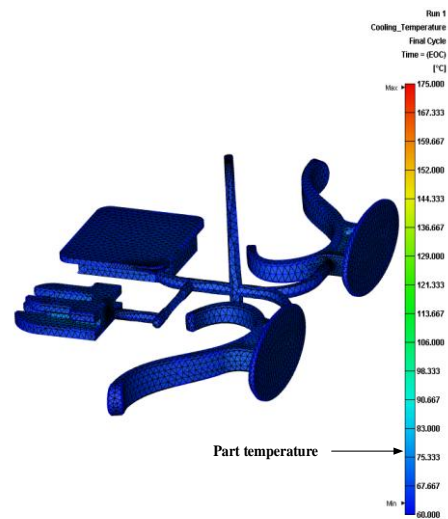


Figure 13. Part temperature at the end of cooling time

Figure 13 shows the temperature of the product after the cooling process, concentrated in the range of 60°C to 75°C. From this, we can see that the cooling system works effectively, and the temperature after the cooling process is evenly distributed across the product, facilitating the ejection of the product.

Additionally, the simulation results also show that the cooling time (Figure 14) is quite short, with the majority of the product's volume being cooled in less than 4 seconds. We can adjust the cooling water flow to change the cooling time accordingly.

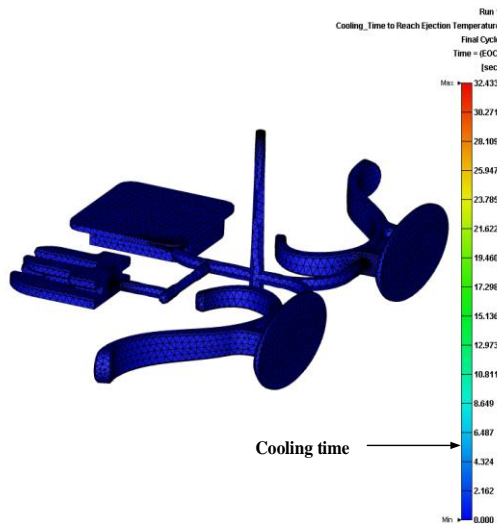


Figure 14. Cooling time

3.3.6. Warpage

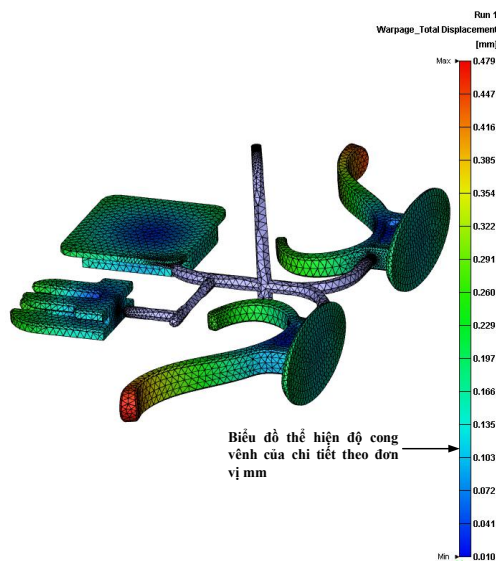


Figure 15. Part warpage

Based on the simulation results in Figure 15, the warpage of the product is 0.47 mm, accounting for 2.29%, which is a relatively small proportion, primarily concentrated at the edges

of the product. Therefore, this warpage does not affect the product.

3.3.7. Weld line

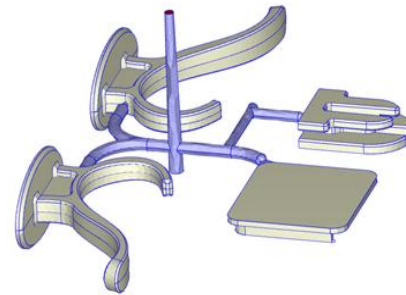


Figure 16. Weld line

Based on the results in Figure 16, we can see that the product after injection molding does not have issues with weld lines, as the plastic flow paths do not intersect, indicating that the injection gate position is quite reasonable.

3.4. Mold manufacturing

3.4.1. Mold design

The mold type designed and constructed by the team in this project is a two-plate mold with an external slider system. This type of mold is widely used due to its simple structure and low cost. To design the mold, we can design directly within the mold separation environment of Creo Parametric. The mold is built and designed according to Misumi standards..

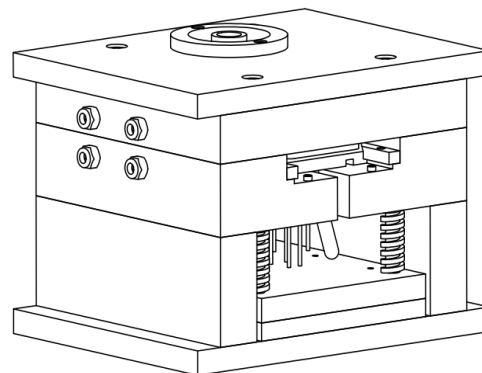


Figure 17. Mold design

3.4.2. Mold manufacturing

CNC milling machines are among the most common machining tools in the manufacturing industry today. Thanks to their ability to process automatically, quickly, and accurately, CNC milling machines have replaced traditional machining tools, helping to increase productivity, improve product quality, and provide flexibility during the machining process. Common milling operations on CNC machines include face milling, slot milling, and pocket milling. In addition, milling machines can also perform drilling, grinding, tapping, chamfering, and more.

To calculate the spindle speed, you can use the cutting speed formula (V_c) to determine the required spindle rotation speed::

$$n = \frac{1000 \times V_c}{\pi \times D} \quad (7)$$

In mold machining, particularly for the die plate, there are four guide pin holes that require high geometric tolerance accuracy. To achieve this, multiple checks are necessary to ensure the precise fit and function of the holes. Additionally, for the ejector pin holes, which also require high geometric tolerance, a smaller drill bit will be used initially, followed by reaming the holes to achieve the most effective precision.

This process ensures that the holes have the required dimensions and finish, reducing errors in the assembly and operation of the mold. The use of reaming after drilling allows for achieving a smoother surface and more accurate hole dimensions.



Figure 18. Core and cavity plates

3.4.3. Mold assembly

a. Cavity plate

Table 2. Part of Cavity plate group

No	Part name
1	Hexagon M6_115
2	Locating Ring
3	Nozzle Bushing
4	Hexagon M10
5	Guide Bushing
6	Cavity Plate
7	Bolts for Ring
8	Taper Pin
9	Lock Block
10	Cavity Insert
11	Water Bushing
12	Hexagon M6x20
13	Hexagon M8
14	Cavity Base Plate

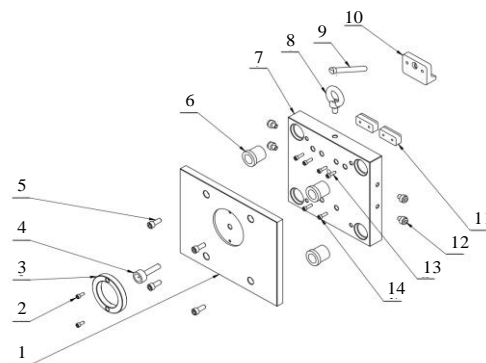


Figure 19. Cavity plate group

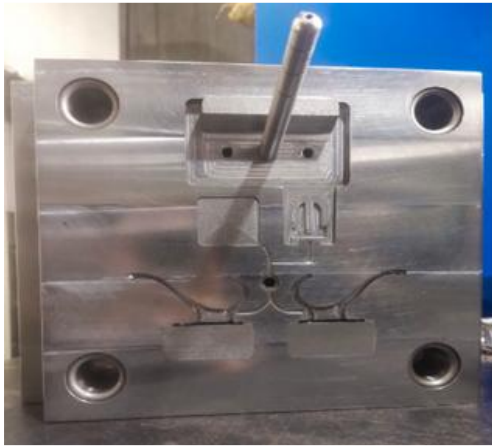


Figure 20. Cavity plate group

b. Core plate group

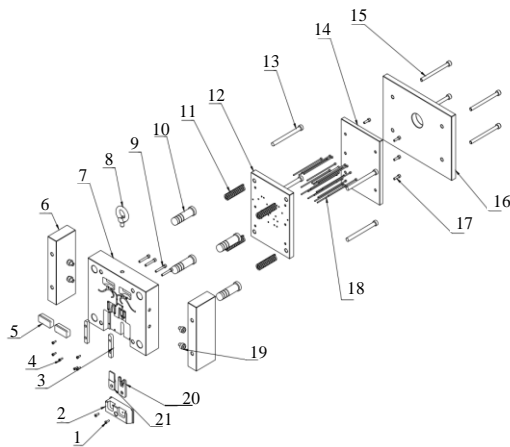


Figure 21. Core plate group

After assembling the cavity and core units, we combine these two parts together to form a complete mold, as shown in Figure 22.



Figure 22. Mold structure

Table 3. Detailed clusters

No	Part name
1	Cavity Base Plate
2	Hexagon M6_115
3	Locating Ring
4	Nozzle Bushing
5	Hexagon M10
6	Guide Bushing
7	Cavity Plate
8	Bolts for Ring
9	Taper Pin
10	Lock Block
11	Cavity Insert
12	Water Bushing
13	Hexagon M6x20
14	Hexagon M8
15	Cavity Base Plate
16	Hexagon M6_115
17	Locating Ring
18	Nozzle Bushing
19	Hexagon M10
20	Guide Bushing
21	Cavity Plate

The injection mold for the wall-mounted plastic hook product demonstrates a high degree of cost-efficiency through its use of a two-plate design with a slider mechanism, which simplifies structure and minimizes material waste. The selection of S50C steel as the mold material offers a cost-effective solution that maintains sufficient hardness and wear resistance for extended use. Production scalability is achieved through the mold's compact dimensions ($320 \times 250 \times 240$ mm) and efficient cycle time—highlighted by a rapid filling time of just 0.216 seconds and optimized cooling time under 4 seconds—allowing for high-volume manufacturing with minimal downtime.

Additionally, simulation via Moldex3D ensures the mold design reduces common defects, further streamlining production. For long-term durability, the mold employs standardized components (per Misumi specifications), precise CNC machining, and high-tolerance fits, which collectively ensure dimensional stability and prolonged mold life under repeated thermal and mechanical loads. Together, these features position the mold as a sustainable solution for consistent product quality, reduced per-unit cost, and adaptability to market demand increases

4. CONCLUSION

The team has successfully designed and manufactured a complete injection mold for wall hooks. The injection mold was designed and processed with high precision, ensuring a perfect fit for the plastic components. The manufacturing process was tightly controlled to ensure that the wall hooks produced from the mold have accurate dimensions and shapes. Through testing by performing trial injections, the mold fully meets the production requirements for the wall hook component, with all technical specifications for the product being met.



Figure 23. Injection molding product



Figure 24. Molding product without sprue



Figure 25. Part in working position

Testing and Adjustments:

The trial molding process has helped the team identify, learn, and adjust technical issues such as the amount of plastic used, injection pressure, and processing temperature to optimize the production process and improve product quality.

With the results obtained, the team is proud of the success of the project in applying technical knowledge and technology to create a product with real-world value and wide application in the market. This project is also an important first step for the team to develop and improve the product in the future, meeting the increasingly stringent requirements of the market and customers.

Although positive results have been achieved, the team faced challenges such as processing time and production costs, leading to the injection mold not being truly perfect. This requires continuous fine-tuning and improvement in the production process to increase efficiency and reduce costs.

To further enhance the value and effectiveness of the product, the team proposes the following development directions:

1. Research and apply more advanced production technologies: The team will continue to study and apply new plastic processing technologies to optimize the production process. Technologies such as 3D printing and high-precision machining will help improve accuracy and production efficiency while reducing material waste.

2. Optimize design to improve production efficiency: The team will continue to develop new designs to optimize the production process, particularly in machining and assembly. Continuous fine-tuning and improvements in design will help reduce production time and costs while enhancing the ability to meet market demand.

REFERENCES

- Ni'am, L. K. M., Budiyanoro, C., & Rahman, M. B. N. (2017). Desain dan Optimasi Injection Mold Sistem Slider pada Produk Stick T15. *Rekayasa Mesin*, 8(3), 155-165.
- Mahajan, L. D., & Ulhe, P. N. (2018). Analysis Of injection molding process to reduced defects (Short-Shot). *International Journal of Engineering Technologies and Management Research*, 5(6), 113-119.
- Masato, D., Sorgato, M., Babenko, M., Whiteside, B., & Lucchetta, G. (2018). Thin-wall injection molding of polystyrene parts with coated and uncoated cavities. *Materials & Design*, 141, 286-295.
- Park, H. S., & Dang, X. P. (2010). Optimization of conformal cooling channels with array of baffles for plastic injection mold. *International Journal of Precision Engineering and Manufacturing*, 11, 879-890.
- Montalvo, J. I., & Hidalgo, M. A. (2015). 3D printing with natural fiber reinforced filament.
- The Uyen, T. M., Truong Giang, N., Do, T. T., Anh Son, T., & Son Minh, P. (2020). External gas-assisted mold temperature control improves weld line quality in the injection molding process. *Materials*, 13(12), 2855.
- Carlson, J. D., Catanzarite, D. M., & St. Clair, K. A. (1996). Commercial magneto-rheological fluid devices. *International Journal of Modern Physics B*, 10(23n24), 2857-2865.
- Yao, D., & Kim, B. (2002). Increasing flow length in thin wall injection molding using a rapidly heated mold. *Polymer-plastics technology and engineering*, 41(5), 819-832.
- Chen, S. C., Liaw, W. L., Su, P. L., & Chung, M. H. (2003). Investigation of mold plate separation in thin-wall injection molding. *Advances in Polymer Technology*:

- Journal of the Polymer Processing Institute, 22(4), 306-319.
- Park, K., Kim, B., & Yao, D. (2006). Numerical simulation for injection molding with a rapidly heated mold, Part I: Flow simulation for thin wall parts. *Polymer-Plastics Technology and Engineering*, 45(8), 897-902.
- Nian, S. C., Wu, C. Y., & Huang, M. S. (2015). Warpage control of thin-walled injection molding using local mold temperatures. *International Communications in Heat and Mass Transfer*, 61, 102-110.
- Song, M. C., Liu, Z., Wang, M. J., Yu, T. M., & Zhao, D. Y. (2007). Research on effects of injection process parameters on the molding process for ultra-thin wall plastic parts. *Journal of Materials Processing Technology*, 187, 668-671.

THIẾT KẾ VÀ CHẾ TẠO KHUÔN PHUN ÉP SẢN PHẨM MÓC NHỰA DÁN TƯỜNG

Trần Duy Nam^{1*}

¹*Trường Đại học Công nghệ Đồng Nai.*

*Tác giả liên hệ: *Trần Duy Nam, tranduydam@dntu.edu.vn.*

THÔNG TIN CHUNG

Ngày nhận bài: 10/01/2025

Ngày nhận bài sửa: 14/04/2025

Ngày duyệt đăng: 03/07/2025

TỪ KHOÁ

Thiết kế;

Mô phỏng;

Chế tạo.

TÓM TẮT

Nghiên cứu này tập trung vào thiết kế và chế tạo bộ khuôn ép nhựa hai tấm có cơ cấu mở slider để tạo ra sản phẩm móc nhựa dán tường. Phương pháp nghiên cứu và thiết kế của chúng tôi dựa trên các tiêu chuẩn và chế tạo để tạo ra một bộ khuôn hoàn chỉnh với kích thước 320 mm × 250 mm × 240 mm, vật liệu làm khuôn là thép S50C. Kích thước của sản phẩm 1 là 72.5 mm × 23 mm × 38 mm, sản phẩm 2 là 37 mm × 35 mm × 6,5 mm, sản phẩm 3 là 29,5 mm × 22 mm × 7,5 mm vật liệu là nhựa PP. Chúng tôi sử dụng các phần mềm Inventor 2024, Creo Parametric 8.0, Moldex 3D để phục vụ cho việc thiết kế, mô phỏng. Kết quả đạt được là thành công tạo ra mô hình khuôn hai tấm có các cơ cấu và chức năng như đã đề ra.