JMPM: Jurnal Material dan Proses Manufaktur
Vol. 9, No. 1, p70, Juni 2025
DOI: https://doi.org/10.18196/jmpm.v9i1.24939

Design Improvement of Mold Bracket for Quarter Trim to Reduce Ejector Mark Defects

Desy Agustin[1], Muhammad Irvan Tantyagus Ma'ruf[2], Edwin Sahrial Solih[3], Fadhil Fadhlurrohman Nurhadi[4], Fredy Sumasto[5]
[1,2,3,4] Automotive Engineering Technology, [5] Automotive Industrial Engineering, Politeknik STMI Jakarta, Jl. Letjen Suprapto No. 26, Cempaka Putih, Jakarta Pusat, DKI Jakarta, 10510, Indonesia
e-mail: desyag@stmi.ac.id[1], irvann1803@gmail.com[2], edwin-solih@kemenperin.go.id[3], fadhilfadhlurrohman@stmi.ac.id[4] , f-sumasto@kemenperin.go.id[5]

Keywords:
Design; Mold; Ejector; Layout; Clamping Force; Pressure

ABSTRACT

This study focuses on improving the quarter trim bracket mold in the mold making division to mitigate the risk of ejector marks, which were identified through long-term analysis. The author redesigned the mold by considering the ejector layout, cooling layout, clamping force calculations, and number of pressure plates, to ensure the product is free from defects. The addition of an ejector pin in the rib area is proposed as a solution to minimize this problem. The quarter trim bracket mold is designed using two pressure plates measuring 30 x 60 mm and four measuring 30 x 80 mm, with a total pressure plate area reaching 13,200 mm². Based on calculations, this part requires a clamping force of 14.5 tons to be produced on a 30 ton capacity injection machine. This clamping force is important to keep the mold tightly closed during the production process, and is calculated based on the cavity pressure in the mold and the shot area. All mold designs are created following company standards to ensure accuracy of sizes and calculations. This research is expected to improve production quality and operational efficiency.

1. INTRODUCTION

In today's manufacturing industry, the need for effective tooling is crucial to support high-quality production processes. Common types of tooling used include molds, dies, and jigs & fixtures, each serving specific functions based on the material being produced. Molds, for example, are used to manufacture products made of plastic [1], while dies are used for sheet metal products [2] [3]. Jigs function as fixtures to hold components securely in place during the manufacturing process [4]. In this context, effective mold design becomes critical to ensure the final product's quality. Design flaws in molds can lead to issues such as ejector marks, which negatively affect the appearance and function of the product [5].

In mold design, there are several key components that play an important role, such as the cavity, core, ejector system, and cooling channels [6]. The cavity and core shape the final product's contours, while the ejector system helps remove the product from the mold after injection [7]. A well-designed ejector system is crucial to minimize the risk of ejector marks, which can occur when the pressure during ejection is uneven [8]. Additionally, the cooling system plays a vital role in ensuring that the plastic material solidifies properly, reducing the risk of deformation that could cause defects.

Proper placement of cooling channels will maintain the mold's temperature at an optimal level, reducing cycle time and improving production efficiency. By considering the harmonious interaction between these components, it is hoped that the risk of ejector marks can be minimized and the final product quality can be enhanced. Previous research used a three-plate mold system, which separates the runner from the product directly when the mold opens. This way, the runner, which was previously connected to the product, can detach without requiring an additional process [5]. The product designed was a glove box, which is a container used for storing gloves. This glove box is made from PP material, which has resistance to both high and low temperatures at room temperature, with the melting point of PP (Polypropylene) being around 230°C [5].

In designing the glove box mold, several aspects need to be considered to minimize product defects, such as the type of runner, gate system, and the mold's cooling system. The analysis in this study of the baffle cooling system showed its effectiveness in maintaining the mold's temperature. From the clamping force calculation, a value of 60.373 tons was obtained. Therefore, the machine used should have a capacity at least 20% higher than this clamping force value [5].

Apart from determining the required clamping force, this research primarily focuses on addressing the risk of ejector marks during the production of the quarter trim bracket mold. Unlike conventional approaches that typically rely on trial-and-error adjustments during molding, this improvement incorporates a systematic analysis of ejector pin layout, material flow behavior, and mold surface conditions. This structured method aims to prevent defects proactively rather than correcting them reactively. Ejector marks often occur due to uneven pressure during ejection, which can lead to increased costs and damage to the brand's reputation. Therefore, it is important to redesign the mold considering the ejector layout, cooling layout, clamping force calculations, and the number of pressure plates. These four aspects are interconnected and significantly influence the mold's effectiveness in the production process. The entire mold design process follows company standards to ensure accuracy and consistency. By adhering to these standards, it is expected that the redesign will reduce the risk of product defects, improve operational efficiency, and contribute significantly to enhanced production quality and provide better insights into mold design for future applications.

2. METHOD

The redesign of the quarter trim bracket mold was carried out due to the long-term issue of ejector marks experienced by the mold. Ejector marks can occur because the position of the ejectors on the quarter trim bracket mold is uneven, and there is no ejector pushing on the rib area, making it more difficult for the part to be ejected from the mold. There are several stages in the mold design process, as shown in the following flowchart:

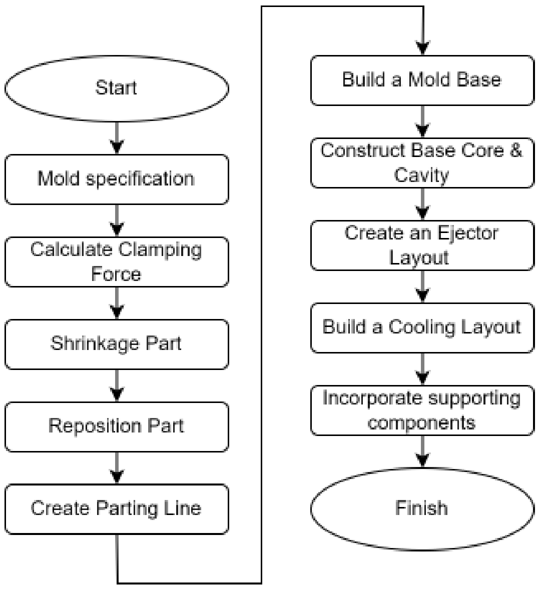


Figure 1. Mold Design Stages

3. RESULTS AND DISCUSSION

3.1 Mold Specifications

The integration of narrow slot feature recognition with automated ejector pin design has been shown to significantly enhance the accuracy of pin placement and overall design efficiency[9] . By analyzing the product’s geometry, ejector pins can be positioned more precisely, reducing reliance on manual adjustments and minimizing the occurrence of surface defects such as ejector marks [9]. This approach has proven effective in improving product quality and serves as a valuable reference for future research in optimizing mold design. In line with this, the current study carries out a mold improvement process that not only considers ejector pin layout but also involves determining essential mold parameters such as clamping force, part weight, and runner type. These mold specifications are crucial to ensure optimal mold performance and part quality, making the overall design approach more integrated and suitable for real manufacturing applications.

3.2. Clamping Force Calculation

Clamping force in the injection molding process refers to the pressure force exerted by the injection machine to hold the mold firmly during the plastic injection process [10] . This force is generated by the machine's clamp unit and aims to ensure that the mold remains tightly closed when molten plastic is sprayed into the mold. This is very important to prevent material leakage and ensure quality product results. The formula for calculating clamping force in the injection molding process is as follows [11]:

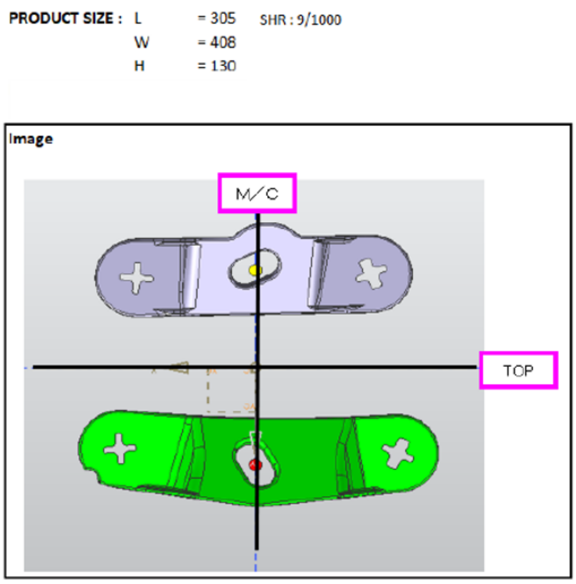
$$F = P \ x \ A \tag{1}$$

Where:

F = is the clamping force required (in Newtons or kN).

P = is the injection pressure (in MPa or N/mm²).

A = is the cross-sectional area of the mold which determines the area where the clamping force acts (in mm²).



PRODUCT AREA (Cm^2)	32,37	the required total clamping force	14567
PRESSURE MATERIAL (kgf/Cm^2)	450	CLAMPING MC. INJECTION	30000
TONNAGE MACHINE	30	CLAMPING CONDITION MUST >1	2,06

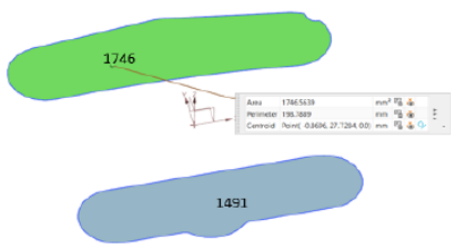


Figure 2. Clamping Force

Based on these calculations, the quarter trim bracket part requires 14.5 tons of clamping so that it can produce on a 30 ton injection machine.

3.3. 3D Part

Apart from the mold specification, 3D parts must also be prepared before starting the mold design, because the 3D parts will be used as a basis for making the mold design. After getting the 3D part, the next step is to calculate the shrinkage and reposition the part based on the direction it was taken when the product was removed from the mold. Apart from repositioning, 3D parts must also be adjusted for shrinkage, namely increasing the size of the part according to the material to be used, to ensure dimensional accuracy of the final product.

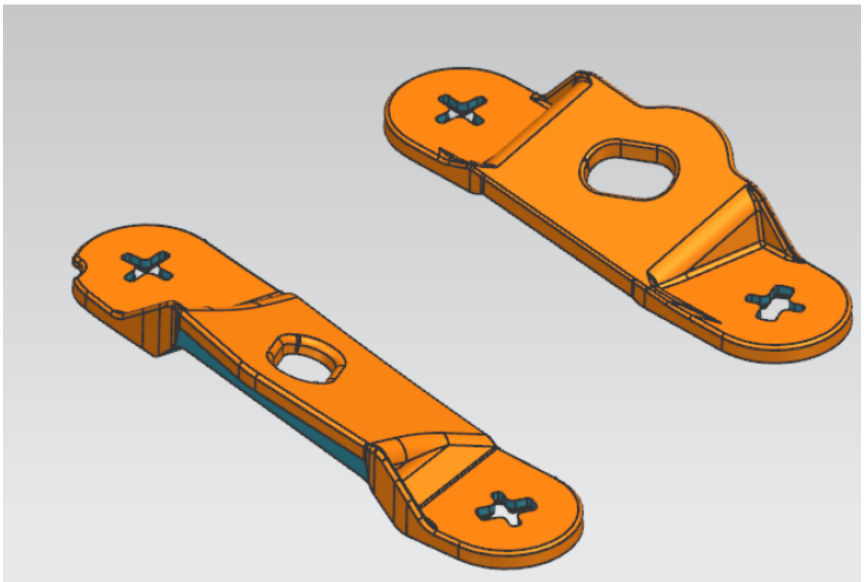


Figure 3. 3D Part

3.4. Part Shrinkage

The purpose of applying shrinkage to the part is to anticipate the shrinkage that will occur during the production process[12] [13]. Parameters such as holding time, injection time, cooling time, and mold temperature are crucial; if any of these injection process parameters are overlooked, the molded part may have defects, such as shrinkage marks [14]. When plastic is heated and then cooled, it tends to shrink, so it is important to understand the amount of shrinkage based on the type of plastic material used. In this case, the plastic material used for the bracket part needs to be analyzed to determine the correct shrinkage coefficient, so that the final product dimensions match the desired specifications. Therefore, the mold design process can be adjusted to accommodate these dimensional changes. Based on TPS, the shrinkage for TSOP-5 plastic material is between 0.0085 and 0.01, so a shrinkage value of 0.009 was used for the quarter trim bracket part, as it still falls within the tolerance limits.

3.5. Part Repositioning

Part repositioning in mold design is a crucial step in the manufacturing process, especially in plastic injection molding. The mold design must also consider the ease of part ejection from the mold after the injection process is complete. Therefore, part repositioning should be done before proceeding with the mold design.

3.6. Creating the Parting Line

The parting line in injection molding is the separation line formed between two opposing mold sections. This line marks the boundary where the cavity and core meet, separating the area that will become part of the final product [15][16].

3.7. Creating Parting Line Draft

Creating a draft for the parting line area can be done using the extrude feature. However, before extruding, ensure that the entire parting line area and the part are merged using the sew feature.

3.8. Creating the Mold Base

After creating the parting line, the next step is to sketch the mold base. This sketch can be used as a reference to create the core and cavity. When selecting a mold base, there are several important considerations to keep in mind [17] [18]:

1. **Outer Area Size**: Ensure that the outer area size of the product in the mold matches the production requirements. This will affect both the efficiency and the quality of the molded product.
2. **Cooling Flow System**: Consider different alternatives for the cooling flow system. Efficient cooling is essential to reduce cycle time and improve the quality of the final product
3. **Opening Molding**: Adjust the molding opening to match the specifications of the injection molding machine that will be used. This is important to ensure that the mold can operate properly and in accordance with the machine's parameters..

The basic size and shape of this mold also refers to the TPS Mold Design Standard. All sizes for the mold base follow the table for the 30 Ton mold because this mold will be produced on a 30 Ton machine. Can be seen in Figure 4 for a basic sketch of the mold suggested by TPS.

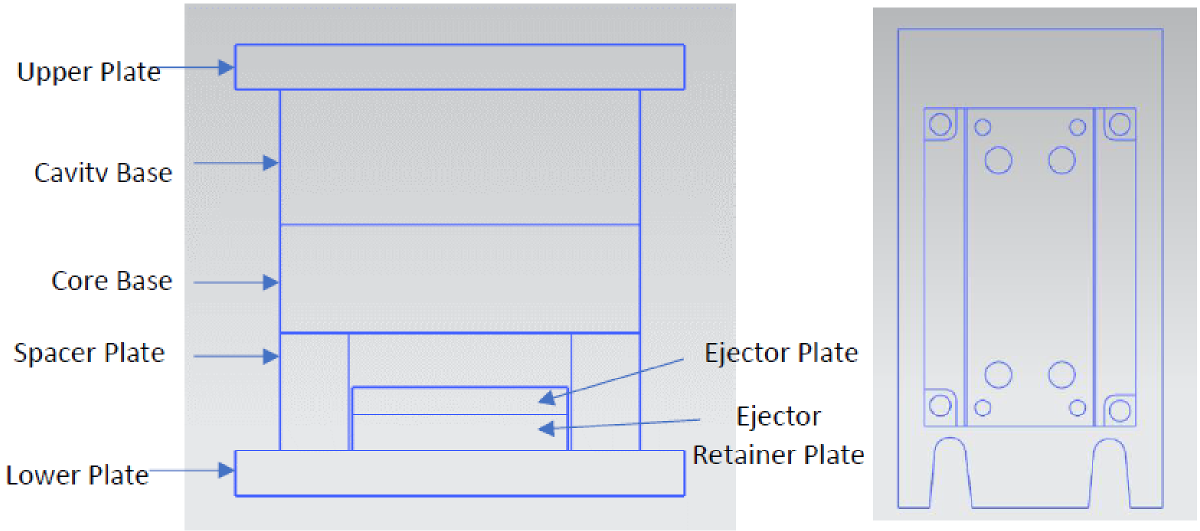


Figure 4. Sketch Mold Base

3.9. Creating Base Core and Cavity

To create the base core and cavity, start by making a cubic shape using the mold base sketch that has been created, as shown in Figure 4. After that, trim the body to ensure the parting line release matches the mold dimensions, as shown in Figures 5 and 6.

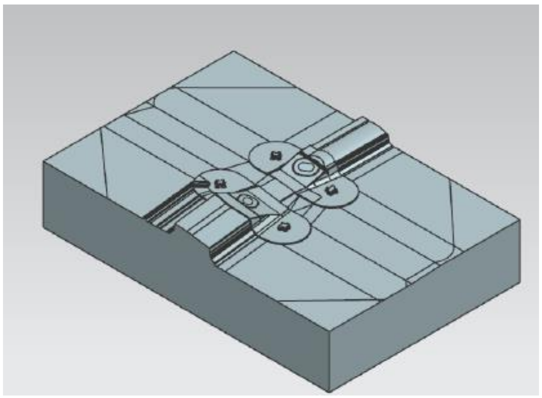


Figure 5. Cavity after body Trim

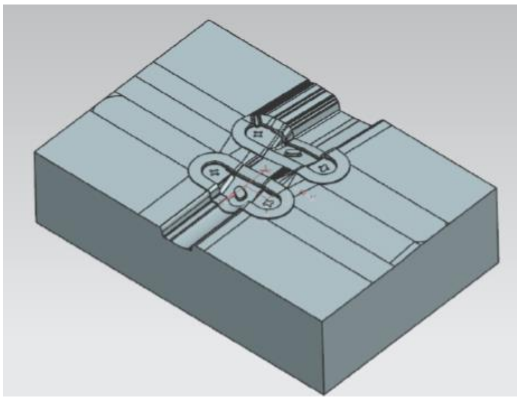


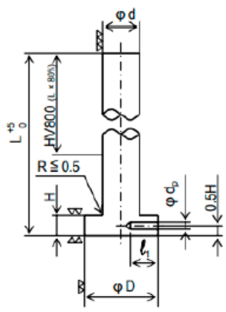
Figure 6. Cavity after body Trim

3.10. Creating the Ejector Layout

When creating the ejector layout, there are several considerations to keep in mind, as follows:

a. Size and Quantity

The number of ejectors used can be adjusted based on the shape of the part. For the quarter trim bracket part, 6 ejector pins are used. The mold for the quarter trim bracket uses Ø4mm ejector pins. The EPN-04 ejector pin is designed with precise dimensional specifications suitable for molds requiring high accuracy, especially in narrow or detailed product areas. The shaft diameter (ϕd) is 4 mm with a tolerance of -0.02 mm to -0.05 mm, ensuring tight fit and smooth movement without jamming during ejection. The head diameter (ϕD) is 8 mm with a tolerance of -0.03 mm, providing sufficient strength and stability under ejector plate pressure. The head height (H) is 3 mm, compatible with standard ejector plate thicknesses. The diameter of the mounting hole (ϕdp) is 3 mm with a tolerance of +0.1 mm / 0, allowing ease of assembly while maintaining positional stability. The minimum straight length (l_1) is 5 mm, ensuring a stable contact surface during product ejection. With these specifications, EPN-04 is an ideal choice for use in molds where small-sized, high-precision ejector pins are required, such as in rib sections or thin-walled product areas. Standard ejector pins can be seen in Figure 7.



	φd	φD	H	φd_p	l_1
EPN-02	$2^{-0.02}_{-0.05}$	$4^{0}_{-0.3}$	$4^{0}_{-0.05}$	$2^{+0.1}_{0}$	3
EPN-03	$3^{-0.02}_{-0.05}$	$6^{0}_{-0.3}$			
EPN-04	$4^{-0.02}_{-0.05}$	$8^{0}_{-0.3}$	$6^{0}_{-0.05}$	$3^{+0.1}_{0}$	5
EPN-06	$6^{-0.02}_{-0.05}$	$10^{0}_{-0.3}$			
EPN-08	$8^{-0.02}_{-0.05}$	$13^{0}_{-0.3}$	$8^{0}_{-0.05}$		
EPN-10	$10^{-0.02}_{-0.05}$	$15^{0}_{-0.3}$			
EPN-12	$12^{-0.02}_{-0.05}$	$17^{0}_{-0.3}$			
EPN-15	$15^{-0.02}_{-0.05}$	$20^{0}_{-0.3}$			
EPN-16	$16^{-0.02}_{-0.05}$	$21^{0}_{-0.3}$			

Figure 6. Standar Ejector Pin

b. Ejector Position

The ejector is an important component in the mold which functions to remove the finished product from the mold cavity after the molding process is complete [8]. Apart from that, the ejector also plays a role in resisting the pressure of the melted plastic material when molding, ensuring that the product is formed well and does not experience deformation. In the molding process, the ejector works by pushing the product out, thereby reducing the risk of damage to the product and the mold itself [19]. Ejector pins are placed around the edges of the product and areas that are not directly visible on the finished product, such as the bottom or sides of the product.

3.11. Creating the Cooling Layout

An effective cooling channel design is crucial. The cooling channels must be designed to distribute cooling evenly around the mold [20]. This includes using channels with the correct diameter, proper spacing between channels, and strategic orientation. The first step in creating the cooling layout is to sketch a circle with a diameter that meets the requirements. For the quarter trim bracket mold, Ø6mm channels are used. Then, extrude the sketch to the appropriate length based on the part dimensions. The Cooling Layout design can be seen in Figure 6.

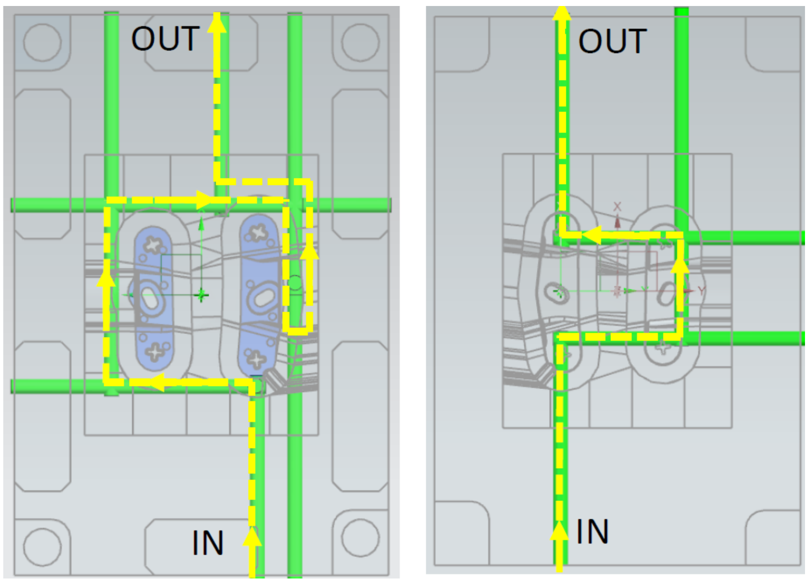
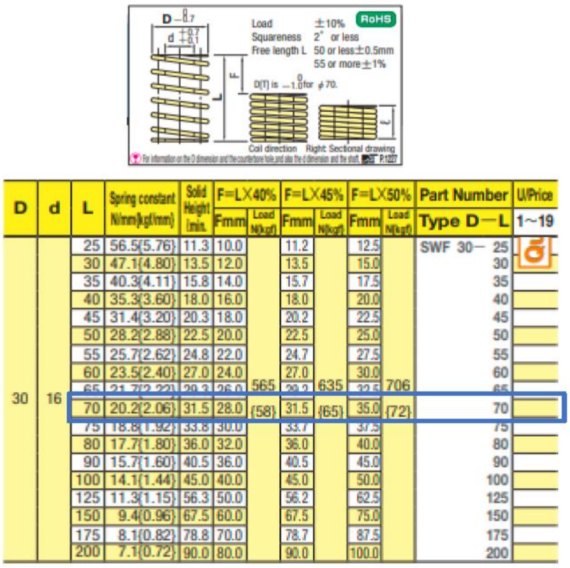


Figure 7. Design Layout Cooling

3.12. Calculate of Spring Ejector

The formula for calculating spring ejectors in injection molding can vary depending on the mold configuration and design. However, in general, the formula used to calculate the plunger spring force is as follows



D	d	L	Spring constant N/mm{kgf/mm}	Solid Height l min.	F=LX40% Fmm	F=LX40% Load N{kgf}	F=LX45% Fmm	F=LX45% Load N{kgf}	F=LX50% Fmm	F=LX50% Load N{kgf}	Part Number Type D—L	U/Price 1~19
30	16	25	56.5{5.76}	11.3	10.0		11.2		12.5		SWF 30— 25	
		30	47.1{4.80}	13.5	12.0		13.5		15.0		30	
		35	40.3{4.11}	15.8	14.0		15.7		17.5		35	
		40	35.3{3.60}	18.0	16.0		18.0		20.0		40	
		45	31.4{3.20}	20.3	18.0		20.2		22.5		45	
		50	28.2{2.88}	22.5	20.0		22.5		25.0		50	
		55	25.7{2.62}	24.8	22.0		24.7		27.5		55	
		60	23.5{2.40}	27.0	24.0		27.0		30.0		60	
		65	21.7{2.22}	29.3	26.0	565	29.2	635	32.5	706	65	
		70	20.2{2.06}	31.5	28.0	{58}	31.5	{65}	35.0	{72}	70	
		75	18.8{1.92}	33.8	30.0		33.7		37.5		75	
		80	17.7{1.80}	36.0	32.0		36.0		40.0		80	
		90	15.7{1.60}	40.5	36.0		40.5		45.0		90	
		100	14.1{1.44}	45.0	40.0		45.0		50.0		100	
		125	11.3{1.15}	56.3	50.0		56.2		62.5		125	
		150	9.4{0.96}	67.5	60.0		67.5		75.0		150	
		175	8.1{0.82}	78.8	70.0		78.7		87.5		175	
		200	7.1{0.72}	90.0	80.0		90.0		100.0		200	

Figure 8. Catalogue Spring

$$F = \ Preload\ x\ Constant \qquad (2)$$

Where:

- F is the force exerted by the plunger spring (in Newtons or kN).
- Spring constant (in N/mm or N/m).
- Preload is the maximum displacement or stroke of the plunger spring (in mm or m)

For the quarter trim bracket mold using spring type SWF 30-70 based on the Misumi catalog which can be seen in figure 9, spring type SWF 30-70 has a constant of 2.06 and a preload of 3mm so the spring can be calculated

$$F = 2{,}006\ x\ 3 = 6{,}18 \qquad (3)$$

Therefore, the SWF 30-70 spring can generate a spring force of 6.18 N under 3 mm of preload compression. This force contributes to the ejection or return mechanism within the mold system to ensure the part is released effectively. In mold design, adequate spring force is essential to overcome friction and adhesion between the molded part and the cavity surface. If the spring force is too low, the part may stick or fail to eject properly, potentially causing cycle delays or defects. The use of the SWF 30-70 spring ensures that the ejector plate or return plate moves with sufficient energy to separate the part from the mold without causing damage or leaving marks.

3.13 Pressure Plate Calculations

The pressure plate on the mold is an important part of the mold which functions to hold and press the mold during the plastic injection process[21]. The main function of the pressure plate is to provide sufficient pressure on the mold and hold the parting line to prevent the parting line from being damaged so that the plastic can be distributed properly and produce quality products. The pressure exerted by the pressure plate must be distributed evenly along the parting line to avoid leaks or defects in the injection product. To determine sufficient pressure to be distributed on the parting line, the surface area of the pressure plate must be greater than the value of the injection pressure minus the area of the parting line or with the following equation:

$$Lp > 1.2 \; x \; Lp/L \quad (4)$$

Where:

- Lp = Pressure Plate Area
- P/L = Parting Line Surface Area

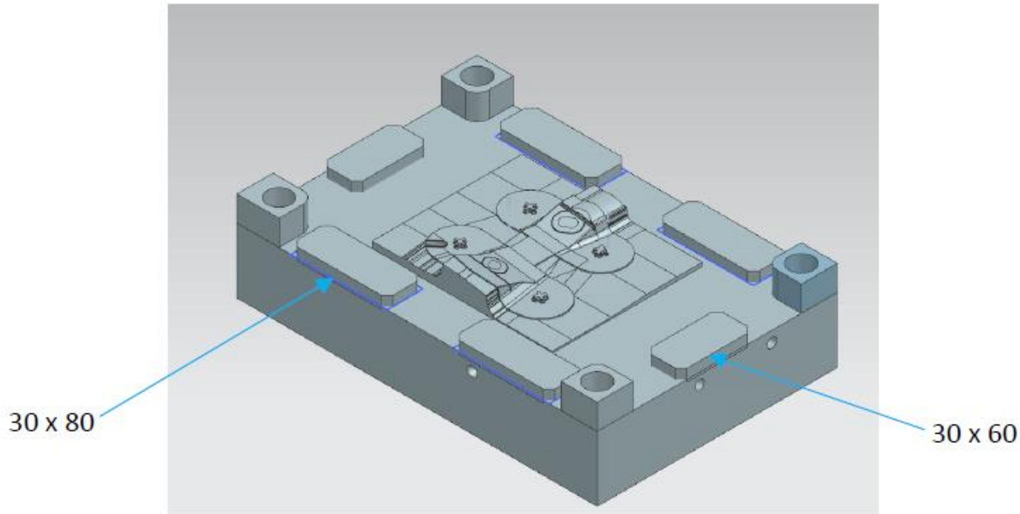


Figure 9. Pressure Plate Position

The Quarter Trim Bracket Mold uses two pressure plates measuring 30 mm × 60 mm and four pressure plates measuring 30 mm × 80 mm. If you add up the area of the pressure plates used in the quarter trim bracket mold is 13,200mm. Meanwhile, the quarter trim bracket mold uses a 30 ton injection machine and has a parting line area of 8,722mm. If you use this equation it will be:

$$13.200 > (0.37 \; x \; 8.722) - (84 \; x \; 30)$$
$$13.200 > (3.227{,}14) - (2520)$$
$$13.200 > (707{,}14)$$
$$18{,}7x \quad (5)$$

Based on this calculation, the inequality holds true, indicating that the clamping force available from the machine (13,200 kgf) is more than sufficient to resist the separation forces acting on the mold during injection. This means that the selected number and size of pressure plates are capable of withstanding the mold opening force and effectively distributing the clamping force across the entire parting line area. Even distribution of this force is critical to prevent flash formation, misalignment, or uneven wear on the mold components. Additionally, ensuring the clamping force exceeds the required minimum helps maintain mold integrity during high-pressure injection and extends the mold's operational life.Based on this equation, the number and size of pressure plates are able to withstand the clamping load and can distribute it evenly to the parting line area.

4. Improvement Design Mold Bracket Quarter Trim

Improvement by adding an ejector pin in the area around the rib as in Figure 11 because there is concern that there will be problems in this area when removing the part. This problem could occur because there is nothing pushing on the rib area so that the rib cannot come off.

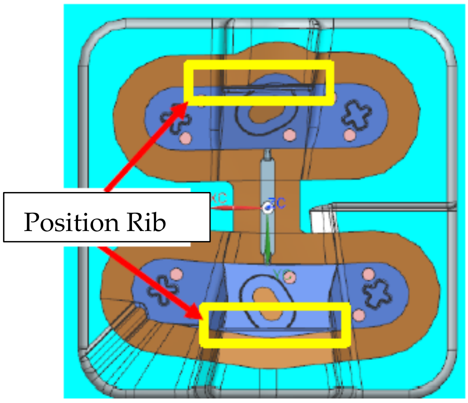


Figure 10. Rib Position Before Re-Design

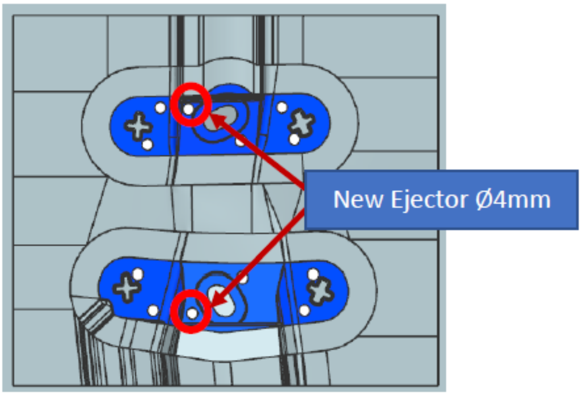


Figure 11. New Ejector Position

There are several possible problems that will occur if you don't add an ejector around the rib area, namely sinkmark problems and ejector mark problems. The sinkmark problem can occur because the part in the rib area is difficult to come out and then it will be pushed forcefully, causing a white mark on the part, while the ejector mark problem occurs because the ejector which is positioned far from the rib area will push the part strongly so that the push of the ejector will leave an impression on it. parts. Both of these problems will cause the part to become NG. In carrying out these improvements, several support pillars had to be removed because the position of the support pillars was right at the position of the newly added ejector, so the support pillars had to be removed.

5. CONCLUSION

Based on the mold design made, it can be concluded:

1. The use of clear mold specifications from the engineering team is very crucial in the mold design process. This helps ensure that all necessary data, such as part weight and runner type, has been properly accounted for to support effective mold manufacturing.
2. Determining the correct clamping force is the key to preventing material leakage during the injection process. Based on calculations, the quarter trim bracket part requires a clamping force of 14.5 tons, which allows the use of an injection machine with a capacity of 30 tons. This shows that accurate counting is essential to maintain product quality.
3. The shrinkage process must be carried out to anticipate plastic material shrinkage during production. Considering the shrinkage value for TSOP-5 material, which is set at 0.009, it is important to ensure that the final part size meets the desired specifications. This shows that material analysis is very important in mold design.
4. Repositioning parts before the mold design process is a crucial step to facilitate product removal from the mold. In addition, creating the right parting line helps separate the mold cavity and core efficiently, minimizing the risk of defects in the final product.

5. Adding ejector pins around the rib area on the mold is very important to prevent problems when removing parts. Without adequate ejectors, the risk of sinkmarks and ejector marks increases, which can result in defective products. Therefore, mold design must consider the position and number of ejectors to improve product quality and production efficiency.

REFERENCES

[1] H. Widiastuti, S. E. Surbakti, F. Restu, M. H. Albana, and I. Saputra, “Identifikasi Cacat Produk Dan Kerusakan Mold Pada Proses Plastic Injection Molding,” *J. Teknol. dan Ris. Terap.*, vol. 1, no. 2, pp. 76–80, 2019, doi: 10.30871/jatra.v1i2.1805.

[2] M. Kurniawan, P. Pardjono, and S. Hadi, “Tingkat Kesiapan Kampus MBKM dalam Bertransformasi Menuju Industri 4.0: Studi Kasus Self Assessment Pusat Pengembangan KKN UNNES,” *J. Penelit. Pendidik.*, vol. 38, no. 1, pp. 1–6, 2021, doi: 10.15294/jpp.v38i1.34212.

[3] A. W. Arohman, D. Agustin, H. Sudrajat, M. Agus, S. Solihin, and M. R. Akbar, “Design and Analysis of Static Stress on Hydraulic Table Dies Replacement,” pp. 1–5, 2023, doi: 10.1109/icced56140.2022.10009919.

[4] D. Agustin, A. W. Arohman, M. Agus, H. Sudrajat, and S. Solihin, “Analisis Finite Element Aluminium Alloy 6063-T5 Pada Jig Positioning Untuk Perakitan Rooftop Interior Mobil,” *J. Ilm. Univ. Batanghari Jambi*, vol. 23, no. 2, p. 1231, 2023, doi: 10.33087/jiubj.v23i2.3654.

[5] A. K. Mufid *et al.*, “Perancangan Injection Molding Dengan Sistem Three Plate Mold Pada Produk Glove Box,” *JMPM J. Mater. dan Proses Manufaktur*, vol. 1, no. 2, pp. 72–81, 2017.

[6] A. B. Wicaksono and A. Surachman, “Pengaruh variasi layout runner dan melt temperature terhadap fill time dan defect pada proses injection molding sikat gigi,” *J. Tek. Mesin Indones.*, vol. 18, no. 1, pp. 38–45, 2023, doi: 10.36289/jtmi.v18i1.425.

[7] P. M. Baterai and L. Led, “Jurnal Ilmiah Setrum,” vol. 8, no. 2, pp. 260–273, 2020.

[8] A. Chandra Maulana, H. Familiana, and C. Suherlan, “Simulasi Pembebanan Statis Pada Ejector Pins Menggunakan Finite Element Method (Fem),” *J. Rekayasa Teknol. Nusa Putra*, vol. 5, no. 1, pp. 22–27, 2021, doi: 10.52005/rekayasa.v5i1.95.

[9] J. Wang, F. Cheng, and Y. Wei, “Research on automatic generation technology of double helix-shaped conformal cooling channels for injection mold,” *Int. J. Adv. Manuf. Technol.*, vol. 134, no. 11–12, pp. 5935–5949, 2024, doi: 10.1007/s00170-024-14545-0.

[10] Siti Qomariah and Agus Dani, “Pengaruh Inject Pressure Dan Clamping Force Terhadap Cacat Flashing Pada Proses Injection Molding Cover Pot,” *J. Energi dan Teknol. Manufaktur*, vol. 6, no. 01, pp. 21–28, 2023, doi: 10.33795/jetm.v6i01.2855.

[11] H. Syaifullah and M. Alfattah, “Pembuatan Alat Bantu Pemesinan Untuk Mempercepat Proses Penghalusan Rib Model X Pada Lini Pemesinan Outer Tube,” *Technologic*, vol. 14, no. 1, 2023, doi: 10.52453/t.v14i1.416.

[12] U. Wahyudi, “Pengaruh Injection Time Dan Backpressure Terhadap Cacat Penyusutan Pada Produk Kemasan Toples Dengan Injection Molding Menggunakan Material Polistyrene,” *J. Tek. Mesin*, vol. 4, no. 3, p. 15, 2017, doi: 10.22441/jtm.v4i3.1269.

[13] A. I. Ramadhan, E. Diniardi, and M. Daroji, “Analisa Penyusutan Produk Plastik di Proses Injection Molding Menggunakan Media Pendingin Cooling Tower dan Udara dengan Material Polypropylene,” *Jrst J. Ris. Sains Dan Teknol.*, vol. 1, no. 2, p. 65, 2017, doi: 10.30595/jrst.v1i2.1577.

[14] A. B. Prasetiyo, F. Fauzun, A. A. Azmi, and S. H. Yaqin, Rizqi Ilmal, Pranoto, “Analisis Keseragaman Pendinginan Produk Plastik Injeksi Molding Dengan Variasi Sistem Pendingin,” *J. Penelit. Saintek*, vol. 25, no. 2, pp. 173–183, 2020, [Online]. Available: https://journal.uny.ac.id/index.php/saintek/article/view/34574

[15] L. Ni’am, C. Budiyantoro, and M. Rahman, “Desain dan Optimasi Injection Mold Sistem Slider pada Produk Stick T15,” *J. Rekayasa Mesin*, vol. 8, no. 3, pp. 155–165, 2017, doi: 10.21776/ub.jrm.2017.008.03.6.

[16] I. Nugraha *et al.*, “Analisa Dan Perancangan Mold Untuk Membuat Mangkuk,” vol. 16, no. 3, pp. 215–219, 2022.

[17] T. Priohutomo, A. Fakharurrozi, T. Mesin, S. Tinggi, and T. Mandala Bandung, “Perancangan Dan Proses Pembuatan Molding Container BKKBN sistem Direct Sprue Gate,” *Isu Teknol. Stt Mandala*, vol. 15 No. 1, no. 1, pp. 12–26, 2020.

[18] R. Hakim, W. Widodo, B. H. Irawan, A. Nurasa, and A. Makruf, “Potensi Novelty Bidang Injection Mold Design: Visualisasi Pemetaan Bibliometric,” *J. Teknol. dan Ris. Terap.*, vol. 2, no. 2, pp. 48–56, 2020, doi: 10.30871/jatra.v2i2.2095.

[19] B. H. Irawan, P. Septaningrum, I. Wijayanti, R. Hakim, and I. Saputra, “Perancangan Dan Modifikasi Mold Insert Untuk Meningkatkan Kapasitas Produksi Light Guide,” *J. Rekayasa Mesin*, vol. 14, no. 3, pp. 927–938, 2023, doi: 10.21776/jrm.v14i3.1457.

[20] F. D. B. Irawan, C. Budiyantoro, and Thoharudin, “Simulasi Desain Cooling System Dan Runner System Untuk Optimasi Kualitas Produk Top Case,” *J. Mater. dan Proses Manufaktur*, vol. 1, no. 2, pp. 63–71, 2017, [Online]. Available: http://journal.umy.ac.id/index.php/jmpm

[21] Y. Yakub and Madinah, “Perencanaan sistem pendinginan cetakan plastik”.