Proceedings of the International Conference on Industrial Engineering and Operations Management

Publisher: IEOM Society International, USA DOI: 10.46254/AF6.20250047
Published: April 8, 2025

Designing and Manufacturing an Efficient Injection Mold

Mohamed Gaber
Senior Student, Manufacturing Engineering Department
German International University, Cairo
Egypt
mohamedramy445@gmail.com

Ahmed Gabr
Teaching assistant, Faculty of Engineering
German International University, Cairo
Egypt
ahmed.gbr2001@gmail.com

Amr Nounou
Associate Professor, Faculty of Engineering
German International University, Cairo
Egypt
amr.nounou@hotmail.com

Abstract

Injection molding is a widely used manufacturing process for producing plastic components with high precision and efficiency. However, optimizing mold design and machining parameters remains a critical challenge to ensure product quality and minimize defects. This study focuses on the optimization of CNC machining parameters for efficient mold design and validation in injection molding. By using CAD/CAM software for mold fabrication and employing Fusion 360 simulations for plastic flow analysis, this research aims to enhance mold performance and production efficiency. The study explores key factors such as material selection, machining strategies, and injection molding conditions to minimize defects like warpage, sink marks, and short shots. Experimental validation through prototype testing and simulation-driven optimization demonstrates significant improvements in mold accuracy and defect reduction. The findings of this research provide valuable insights into cost-effective and high-precision mold manufacturing, making it particularly relevant for small-scale production using desktop injection molding machines.

Keywords
Mold, Injection, CNC, Optimization, Machining

1. Introduction
Plastic products became very common in modern life. It started replacing metal products specifically because it is lighter in weight, less expensive, more available, and easier to shape. These advantages lead to 70 percent of consumer products being produced by the polymer production business. One of the techniques of creating a plastic product is plastic injection molding, which is injecting melted plastic into a mold that has the shape of the required product. The injection molding process has become an indispensable tool in modern manufacturing, enabling the efficient

production of a vast array of plastic products. However, the success of this process hinges on the quality of the molds used. The design and manufacturing of these molds are critical factors that influence the overall efficiency, cost-effectiveness, and product quality of the injection molding process.

1.1 Objectives

One of the types of injection molding machines is the desktop injection machine, which has the same structure as a regular injection machine but with smaller scale. Limited sized molds are applied to this machine producing limited sized products. The Aim of this study is to produce a larger and more complex product using the desktop injection machine by optimizing the product's mold design with overcoming the machine's limitations.

2. Literature Review

2.1 CNC Machining Optimization

The study by Ibrahim I Ikhries and Ali F Al-Shawabkeh (2024) conducted a comprehensive study to optimize CNC machining parameters for the production of complex mold inserts. They employed a Taguchi method to identify the most influential parameters and determine their optimal levels. The results demonstrated that by carefully adjusting cutting speed, feed rate, and coolant flow rate, it is possible to achieve significant reductions in machining time and improve surface quality. The study by Robert Kelley Bradley (2022) explored the effects of cutting speed and feed rate on surface finish and dimensional accuracy of mold cavities, finding that optimized parameter settings can significantly improve part quality.

2.2 Moldflow Optimization

The study by Satadru Kashyap and Dilip Datta (2015) utilized Moldflow to investigate the effects of mold temperature, injection pressure, and cooling time on part warpage and residual stresses. They developed an optimization framework based on genetic algorithms to determine the optimal parameter settings that minimized part distortion and improved dimensional accuracy.The study by I Santos, AJ Pontes, and CI Martins (2014) used the same techniques but focused on optimizing the gate location in injection molds using Moldflow. They compared different gate configurations and runner layouts to evaluate their impact on flow distribution, filling time, and part quality. The results highlighted the importance of careful gate placement in preventing defects and increasing cycle times.

2.3 Material Selection

The study by AT Machining (2024) studied the impact of mold material on part quality and cycle time in injection molding. They compared different mold steels and found that materials with higher thermal conductivity can reduce cooling time and improve part dimensional accuracy. The features of the mold materials selected was according to how much it was relevant to the thesis, for example: cost efficiency and cycle time, other features like weldability of the material were not relevant. The material that fulfilled these requirements was aluminum. Compared to steel molds, aluminum injection molds perform significantly better at eliminating heat from the material. They can, therefore, quickly heat up and cool down.

The study by Gabriel Antonio Mendible (2022) found that materials with high fluidity, and lower temperature and pressure requirements are more suitable for aluminum molds as aluminum has inferior mechanical properties and is less capable of withstanding high-injection speed and high-pressure conditions. This limitation can lead to flashes even in new molds. Therefore, aluminum molds are best suited for applications involving simple product designs. Polypropylene (PP) was the best polymer to be used in an aluminum mold as it has a lower melting point compared to many other thermoplastics, which makes it easier to process and reduces the thermal stress on aluminum molds. PP also has excellent flow characteristics, allowing it to fill intricate mold designs effectively, which is particularly beneficial for aluminum molds that may have complex shapes. while materials like PC and glass fiber-reinforced plastics are less compatible due to their higher temperature and pressure demands.

3.Methodology

3.1 Describing the manufacturing sequence

To establish a comprehensive understanding of the existing injection molding processes, a thorough literature review was conducted. This review covered various aspects of mold design, CNC machining, and injection molding, including traditional methods, recent advancements, and best practices. The objective was to identify potential areas for optimization and gain insights into the challenges and opportunities associated with the manufacturing processes.

3.1.1 Literature Review and Preliminary Research

A comprehensive literature review was conducted to explore existing research on mold design, CNC machining, and injection molding. This review focused on identifying relevant studies, analyzing methodologies, and evaluating the findings to establish a baseline understanding of the field. Additionally, preliminary research was undertaken to gather specific information about the injection molding processes and equipment used within the organization.

Identifying Current Planning Practices

The current planning practices employed in the organization were carefully examined to identify potential areas for improvement. This involved analyzing the existing processes for mold design, CNC machining, and injection molding, including the selection of materials, parameter settings, and manufacturing sequences. The goal was to identify any inefficiencies or bottlenecks that could be addressed through optimization. The plastic product needs to be selected so candidate options are compared to use the most suitable product to be injected. The product needs to be small in size and usable in many applications.

3.2 CAD/CAM Optimization

CAD/CAM software was utilized to optimize the design and manufacturing of molds. By carefully adjusting CNC machining parameters, such as cutting speed, feed rate, and depth of cut, it was possible to improve machining efficiency, reduce tool wear, and enhance part quality. Additionally, CAD/CAM tools were employed to optimize mold geometry, ensuring proper flow paths and minimizing potential defects.

Plastic Flow Simulation

Fusion 360 simulation software was employed to predict the behavior of plastic materials during the injection molding process. By simulating factors such as temperature, pressure, and flow rate, it was possible to identify potential defects, optimize process parameters, and improve part quality. Fusion 360 simulations were used to evaluate different mold designs and manufacturing sequences to select the most promising options.

Implementation and Testing

Optimized mold designs and manufacturing sequences were implemented in the organization's injection molding machines. The performance of the new processes was carefully monitored and evaluated through rigorous testing. This included measuring part quality, cycle time, and overall efficiency. Data was collected and analyzed to assess the effectiveness of the optimization efforts.

3.3 Data Analysis and Reporting

The collected data was analyzed using statistical methods to identify trends, correlations, and significant differences. The results were then summarized and presented in a clear and concise manner. The report included recommendations for future improvements and potential areas for further optimization.

Implementation

The selected product to be injected is a powder scoop, since it has less complex geometry and consisting of two components which are the handle and the scooping unit and each one has different thicknesses, which creates a proper challenge while designing and injecting, leaving space for optimization. Based on the machine limitations and the simulations in the results section an optimization of the product design and the mold design will be made to ensure having a high quality product with minimum amount of defects.

There are numerous steps involved to establish a proper design capable of injection, f irst the product dimensions and geometry needed to be optimized to match the lowest pressure possible. Then the product weight needs to me measured as it needs to be below 10 grams. Then the volume of the scooping unit needs to be standardized for practical uses. After reaching a proper design of the scoop using these parameters, a mold design needed to be optimized to endure the stresses faced from clamping and injection. Then making sure the design is easy to be machined by reducing the number of faces, after that applying the ejection system and then apply the design for assembly

3.3.1 Designing The Product

A total of ten scoop designs are made on solidworks CAD software to reach the optimum design, the designs are divided into 4 groups Initial designs, Bowl designs, Shovel designs and Final designs. After each design the design is uploaded on Fusion 360 software to conduct the injection pressure simulation then the scooping capacity is checked for usability. To know the machine's injection pressure a previously injected product which is a plastic coaster was reverse engineered and unloaded on fusion 360 to observe its injected pressure, the observed injection pressure was 12.7 MPa. Each injection pressure for each scoop design is compared to the 12.7 MPa injection pressure, if the injection pressure in lower than 12.7 MPa and the scooping capacity is usable then the scoop design is acceptable if not, a design iteration is needed until reaching the optimum design.

3.3.2 Initial Design

The initial design is made based on the basic structure of an ordinary powder plastic scoop consisting of two main parts the scooping unit and the scoop's handle. The scoop's capacity was acceptable which is 20 ml but the injection pressure was 15.8 MPa which was higher than the reference point. To address these challenges, the hole was removed, and the groove was replaced with separate circular grooves. Additionally, sharp corners were replaced with fillets to improve part quality and mold performance. By reducing the complexity of the part geometry, the mold could be manufactured more easily, and the injection molding process could be more efficient and reliable. But the pressure only reduced by 0.2 MPa.

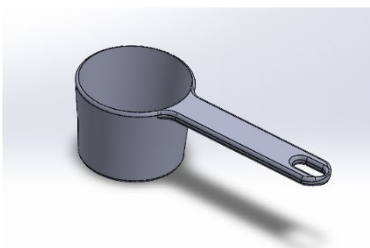

Figure 1. Initial Scoop Design

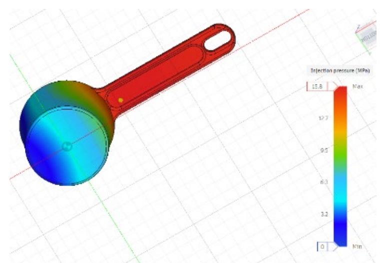

Figure 2. Initial Design Injection Pressure Simulation

Bowl Design

The bowl part of the scoop needed the most pressure so it needed to be simplified so to minimize the pressure, the volume of the bowl part was reduced and the length of the handle was reduced. After simulation the design didn't satisfy the needed pressure so further designs were made to decrease the bowl volume and thickness Low pressure was maintained according to the simulations but the scooping volume was too small and not applicable, so another design iteration had to be done to the scoop.

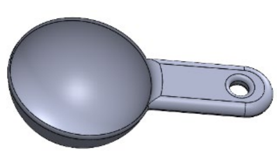

Figure 3. Bowl Design

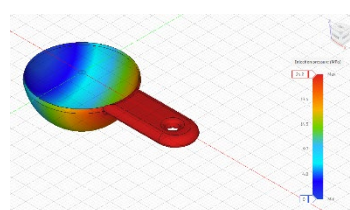

Figure 4. Bowl design injection pressure simulation

Shovel Design

An alteration in the structure of the design was an approach to make sure the design secure the lowest pressure with the most scooping capacity. The scooping unit changed from a tapered cylindrical shape to a Shovel shape, maintaining the main function which is scooping. The scoop injection pressure was acceptable but the scooping capacity needs to be increased.

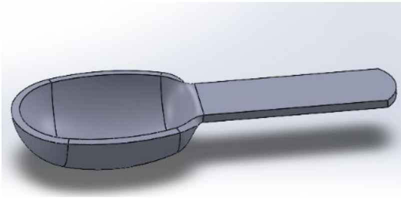

Figure 5. Shovel design

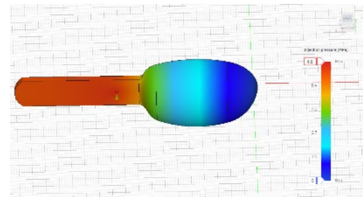

Figure 6. shovel scoop injection pressure

3.3. 3 Final Design

The scooping capacity needed to be increase to be more applicable and to be used in more applications but at a certain pressure limit which is less than 12.7 Mpa. The first design has a capacity of 10 ml with a 6.7 Mpa pressure, The second design has the first structure shape of the initial design with a scooping capacity of 15 ml which is more usable.

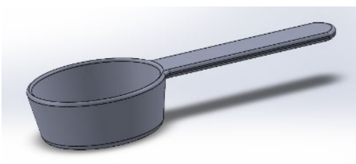

Figure 7. Final scoop design

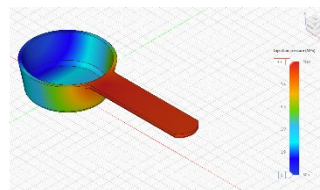

Figure 8. Final design injection pressure

3.3.4 Prototyping

Prototyping before injection molding offers numerous advantages, enabling the validation of design and functionality, while identifying and resolving potential issues early in the development process. It helps reduce costs by preventing expensive mold rework. Additionally, prototyping mitigates risks by uncovering manufacturing challenges, optimizing mold flow and gate placement. The Final design was 3D printed to physically the reliability of the product.

Figure 9. Final design prototype

3.4 Mold Design

The mold design is the most critical step in the implementations. Aluminum was selected as the material for the mold, as aluminum is easier to machine and has a fast cooling ability so cooling channels will not be needed as the mold have a small size as well. There are some factors that were taken in consideration while designing the mold. First of all the mold cannot exceed a specific thickness as the machine has an opening stroke of 150 mm. Secondly, the mold needs to be perfectly aligned to avoid core shift and to make sure that the product has the required thickness. Then all of the core face must touch all of the cavity face to avoid flashes. then ensure that all the features of the mold can be machined easily. In the end ensure that there are no small thicknesses in the mold to avoid fatigue stress and cracks while clamping.

The scoop design is imported into SolidWorks CAD software, where it is scaled to account for the material shrinkage factor, ensuring the final product meets the required dimensions after molding. Polypropylene typically exhibits a shrinkage factor ranging from 1 percent to 3 percent, as outlined by Specialchem (2024),. For this design, a 2 percent shrinkage factor was selected, resulting in a scale factor of 1.02 applied to the original dimensions of the scoop. A parting line is defined on the scoop to separate the core from the cavity. This line is used to establish a plane, from which the core and cavity are extruded to form a cube-shaped mold. The core thickness is set to be 65 mm and the cavity has a thickness of 22 mm. After that a groove is extruded from the core for ejection system

Air vents are applied to the cavity of the mold which is a groove made at the two sides of the cavity, it is made to get rid of the trapped air in the mold while injection to prevent burn marks and short shots as mentioned in the background chapter 2. The air vent has a width of 3mm and a depth of 0.04mm.

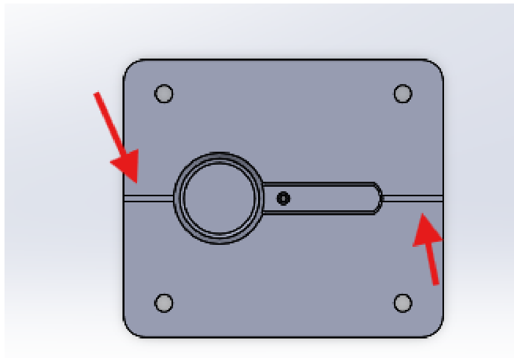

Figure 10. Air Vents

3.5 Feeding System

The feeding system is designed in the cavity section. It is designing the path for the molted material inside the cavity from the nozzle of the machine to the end of the cavity where the part is formed. There are two candidates for feeding systems, Gate runner system and direct sprue gate system. According to Basilius (2024), Xcentric (2024) and Mehdi Moayyedian and Kazem Abhary a comparison between the gate runner system and direct sprue gate system is done in Table 1.

Table 1. Features comparison between gate runner system and direct sprue system

Feature	Gate runner system	Direct sprue gating
Material waste	High	Low
Tooling cost	Higher	Lower
Cycle Time	Longer (due to the runner cooling)	Lower
Design Flexibility	High (suitable for complex and multi-cavity molds)	Limited (best for simple, single-cavity parts)
Part Quality	consistent	Low (gate mark is visible)
Material Efficiency	Moderate (due to runner waste)	High (direct filling, no waste from runners)

Based on this information the direct sprue gating is selected as the feeding system in the mold and the main item to be designed id the sprue. according to Firstmold (2024), The sprue needs to have a coned shape.

3.6 Mold Accessories

There are some accessories that is added to the mold that have a fixed structure and not designed, and there are accessories that is designed which is the ejection system. The ejection system is responsible for ejecting the plastic product after the injection process. It is located in the core side and consists of upper ejector plate, lower ejector plates, ejector pins and return pins. The ejector plates are responsible for moving the ejector system, the ejector pins are located on the ejector plates and responsible for ejecting the product and the return pins are located on the ejector plates and responsible for relocating the ejector system back while injection after the ejection process.

Aligning pins are implemented on the core side each pin has a corresponding hole in the cavity part. It is responsible for aligning the core and cavity together while clamping. A sprue bushing is applied in the back of the cavity, it is responsible for transferring the molten plastic from the machines' nozzle to the mold as it has the sprue design in it.

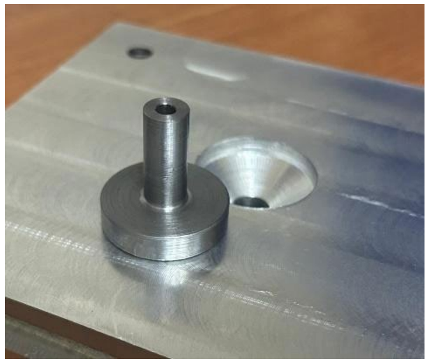

Figure 11. Sprue bushing

3.7 Design For Assembly

The assembly of the mold is a critical step, especially considering the machine's limited stroke of 150 mm. An existing mold mounted on the machine was disassembled and analyzed. In the old mold design, the screws were located on the inner faces of the core and cavity, making assembly and disassembly difficult. This design required the person handling the mold to fit their hand into the narrow gap between the core and cavity, which was both challenging and time-consuming

To make the mounting of the mold on the machine easier and more flexible a slot is made along the core and the cavity sides, then after the mold being mounted and aligned on the machine plates disposable aluminum clips are installed on the slots pushing each side of the mold towards each plate by using screws. This method will eliminate the step of designing holes on the mold that must align precisely with the clamping plate.

3.8 CAM and Machine's Parameters

The CAM program code is divided into four parts the core, the cavity, upper ejector plate and lower ejector plate. The cavity has the most complex geometry and the most features to machine. In this section steps of machining on Fusion 360 CAM simulation will be illustrated. The Material of the tool selected is carbide as they are preferred for their wear resistance, high-speed capability, and precision. After selecting all the CAM operation, a G-code is generated by post processing and then applied to the milling machine

First of all The part is uploaded to the software, then they 3 axis of the machine is set making the z axis perpendicular to the part surface after that the stock offset is set according to the available material. The side and top offset is set to 1 mm each. Each part passes through roughing and finishing operations for example, the cavity part passed through 8 CAM operations which are:

1. **Facing:** removes the excess 1 mm from the face of the part. It removes surface irregularities, creating a reference plane for accurate alignment and precision in multi-step processes.
2. **2D Contour:** removing the excess material from the sides of the part to reach the dimensions of the designed part.
3. **2D Pocket:** a roughing step made to remove the material from pockets that have small thickness as the handle cavity.
4. **Adaptive clearing:** a roughing operation in which an excess amount of material is removed from the part for example: the scooping unit cavity in the cavity part and some parts in the handle cavity.
5. **Drilling Operation:** four holes made for aligning pins and one hole for the sprue bushing. Drills are made with a 3mm spot drill with a flute length of 45 mm to guide the boring tools and increase the boring tool life.
6. **Boring Operation:** The five holes that has been drilled in the drilling operation are shaped by a 5 mm ball end mill with a flute length of 45 mm.

7. **Contouring:** a semi finishing operation for steep areas. In this operation chamfers are made on the 4 aligning holes, air vents are shaped, fillets on the hand cavity are made and the scooping unit is finished. This operation is made by a 5 mm ball end mill with a 45 mm f lute length.
8. **Scallop Operation:** used for finishing all the part by using a 5 mm diameter ball end mill with a flute length of 45 mm.

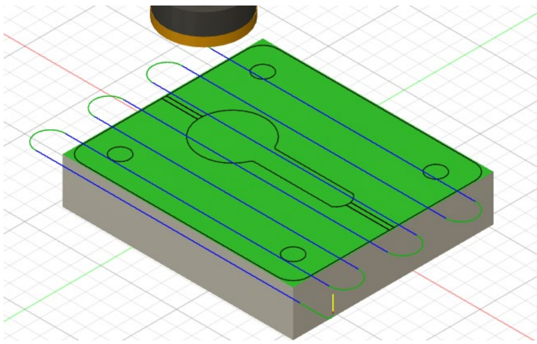

Figure 12. Facing operation

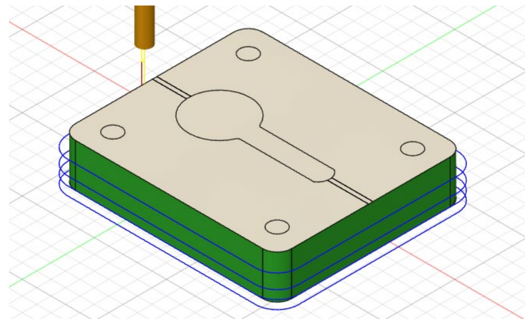

Figure 13. 2D contour operation

Injection Parameters

Improving injection molding cycle time doesn't just rely on mold design; optimizing the injection parameters also plays a key role in reducing cycle times. Injection pressure can be controlled by two main injection parameters which are mold temperature and melt temperature. For a melt temperature range of 180°C to 260°C, the most commonly selected temperature is 220°C. The mold surface temperature typically ranges from 20°C to 80°C. Since the machine does not provide mold heaters, a mold surface temperature of 50°C has been chosen, which aligns with the recommendations from the simulation software.

4. Results & Discussion

Results have two parts software results and hardware results. The software results are the results of the simulation on the implemented design based on the applied machine working with average or high performance. The hardware results are the machining the mold using the CAM parameters in the implementation section then testing the quality of the surface finish then mounting the mold on the machine and test the assembly and alignment. Then the injection molding results (Final product)

Design iterations are based on injection pressure, with the scoop's structural design being the primary influencing factor. Three additional parameters that slightly affect injection pressure are melt temperature, mold surface temperature, and injection point placement.There are two injection points that can be selected in the scoop design either on the hand of the scoop or at the bottom of the scooping unit. Ten designs are made in the implementation chapter based on the injection pressure which is measured at the two injection points as shown in the following Table 2. The first injection point is at the hand of the scoop and the second injection point is at the bottom of the scooping unit.

Table 2. Injection pressure results at two different injection points

Design	Capacity (ml)	Injection Pressure at 1st point (MPa)	Injection pressure at 2nd point(MPa)
Initial scoop design	20	15.8	18.8
Second scoop design	20	15.6	17.4
Bowl scoop	5.5	24.8	22.3
2nd bowl scoop	7	12.6	12.2
3rd bowl scoop	1.3	17.1	13.2
shovel scoop with holes	8	11.5	12.2
shovel scoop with no holes	8	6.8	7.4
10 ml scoop	10	6.7	8.9
15 ml scoop with holes	15	10.3	11.5
15 ml scoop	15	9.3	10.1

Observations: There are two observations in Table 3 . The first observation is that the first injection point provides a lower injection pressure except in the first and second bowl scoops. The second observation is eliminating the holes in shovel scoop and 15 ml scoop reduced the injection pressure by 4.7 MPa and 1 MPa respectively.

Conclusions:

1. The first injection point is the optimum point for injection
2. Eliminating the holes and grooves from the design results in a lower injection pressure as the flow of the molten plastic in the mold will flow smoothly without interruptions.
3. The 15 ml capacity and 9.3 MPa injection pressure are the most suitable results across the scoop designs

After selecting the 15 ml to be injected further optimization is made to reach a lower pressure, this is made by changing the melt temperature and mold temperature in simulations and observe the injection pressure response to these changes.

Table 3. Injection pressure of 15 ml scoop at different melt and mold temperatures

Melt Temperature (°C)	Mold Temperature (°C)	Injection Pressure (MPa)
220	50	9.3
220	30	9.6
240	50	8.0
240	30	9.0

Observations:

1. At melt temperature of 240°C and mold temperature of 50 the part has the lowest injection pressure
2. The part has a higher injection pressure after decreasing the mold temperature to 30°C

Conclusions:

1. As melt temperature increase the injection pressure decreases because the molten plastic in the sprue solidifies slower than usual so less injection pressure is needed.
2. As mold temperature increase the injection pressure decreases because the wall of the mold keeps the material molten for a longer time which makes the flow easier and reduces the shear rate.

Each part in the mold is manufactured and assembled, the core has a thickness of 60 mm while the cavity has a thickness of 30 mm leaving a 60 mm gap between the core and the cavity while ejection of the part.

Figure 14. Finished mold (core and cavity)

Figure 15. Mold mounted on the machine

The final product conducted as the required design.

Figure 16. Final products

Figure 17. Sample of final product

a. Proposed Improvements

- High grade aluminum material was not available at the time of implementation. Using a higher-grade aluminum as 7075 will increase the mold life and reduce product defects.
- Material with high heat conductivity is required for the sprue to prevent the molten plastic from fast solidification preventing the rest of the material from reaching the rest of the cavity.
- Heat treatment enhances the mold's mechanical properties, such as hardness, wear resistance, and fatigue strength, which are critical for molds that are subjected to continuous injection molding cycles.

5. Conclusions

This thesis demonstrates a systematic approach to optimizing CNC machining parameters and mold design processes to enhance efficiency, reduce production costs, and improve the quality of plastic injection-molded products. By integrating CAD/CAM software for machining optimization, Plastic flow simulations for refining injection parameters, and strategic material selection, significant advancements were achieved in reducing machining time, enhancing surface finish, and minimizing defects like flash, sink marks, and air pockets. Aluminum molds proved advantageous for heat dissipation, cost efficiency, and faster manufacturing, though challenges with durability and surface finish remain. The findings underscore the value of precise machining adjustments, optimal temperature and pressure conditions, and the potential for improved performance through advanced heat treatments and innovative materials. By addressing limitations and overcoming constraints, this study lays a foundation for future breakthroughs in mold design and manufacturing, providing a robust methodology for driving innovation and achieving high-quality, cost-effective production.

References

Basilius, B. and Bronkhorst, B., *Types of gates for injection molding*, 2024.

Bradley, R. K., *Education in plastics manufacturing: Aluminum mold making and injection molding*, *International Journal of Mechanical Engineering Education*, vol. 50, no. 3, pp. 726–738, 2022.

PCS Company, *Sprue bushing B-600-134*, 2024. Available: https://www.pcs-company.com/

Ikhries, I. I. and Al-Shawabkeh, A. F., *Novel methods for optimizing CNC aluminum alloy machining parameters in polymer mold cavities*, *International Journal of Lightweight Materials and Manufacture*, vol. 7, no. 4, pp. 507–519, 2024.

Kashyap, S. and Datta, D., *Process parameter optimization of plastic injection molding: A review*, *International Journal of Plastics Technology*, vol. 19, no. 1, pp. 1–18, 2015.

AT Machining, *Aluminum injection molds in plastic*, 2024. From https://at-machining.com/

Boyan Manufacturing, *Aluminum injection molds: Pros, cons, and choice*, 2024. From: https://boyanmfg.com/find-a-good-plastic-mold-manufacturer-in-china/

Mendible, G. A., Saleh, N., Barry, C., and Johnston, S. P., *Mechanical properties and crystallinity of polypropylene injection molded in polyjet and aluminum tooling*, *Rapid Prototyping Journal*, vol. 28, no. 4, pp. 686–694, 2022.

Moayyedian, M., Abhary, K., and Marian, R., *Gate design and filling process analysis of the cavity in injection molding process*, *Advances in Manufacturing*, vol. 4, no. 2, pp. 123–133, 2016.

Xcentric Mold, *Gates and runners*, 2024. Available: https://xcentricmold.com/.

Ozcelik, B., Ozbay, A., and Demirbas, E., *Influence of injection parameters and mold materials on mechanical properties of ABS in plastic injection molding*, *International Communications in Heat and Mass Transfer*, vol. 37, no. 9, pp. 1359–1365, 2010.

Santos, I., Pontes, A. J., and Martins, C. I., *Optimization of injection molding process of polypropylene with aluminum metallic pigments*, *Institute for Polymer and Composites*, Madrid, Spain, 2014.

SpecialChem, *Polymer property: Shrinkage*, 2024. Available: https://www.specialchem.com/

Biographies

Mohamed Gaber is currently pursuing his B.Sc. degree in Manufacturing Engineering at the German International University (GIU) in Cairo, Egypt, Mohamed's research focuses on optimizing injection mold designs to produce products that can be applicable with any machine and to overcome any limitations

Ahmed Gabr is currently pursuing his M.Sc. degree in Manufacturing Engineering at the German International University (GIU) in Cairo, Egypt, where he also serves as a Graduate Teaching Assistant. In this role, he is actively involved in teaching and guiding students through their academic journey in engineering. Ahmed's research interests focus on material handling systems design and analysis, where he is dedicated to optimizing manufacturing processes and improving production efficiency. He aims to leverage advanced technologies and innovative methodologies to address challenges in the field, contributing to the development of effective and sustainable engineering solutions.

Amr Nounou Amr Nounou is currently Associate Professor at the Faculty of Engineering at the German International University (GIU) in Cairo, Egypt. He obtained Dr.-Ing. in Mechanical Engineering as well as M.Sc. in Textile and Clothing Engineering (at the faculty of Mechanical Engineering) from Dresden University of Technology (TUD) and B.Sc. in Mechanical Engineering from The American University in Cairo (AUC). His research interests are in the areas of lean manufacturing, quality improvement, thermal ergonomics, facilities planning and strategic management. He has more than ten years of experience in the industry in the fields of production management, product development, project management, change management and strategic planning