الجـــــــــــــــــــــمهوريـــــة الجـــــــــــــزائريـــة الديمـــــــــقراطيـــة الـشـــــــــعبيـــة

PEOPLE'S DEMOCRATIC REPUBLIC OF ALGERIA

وزارة التــــــــــــــــــــــــــعـليــم العـــــــالي والبـــــــــــحث العـــلــمـــي

Ministry of Higher Education and Scientific Research

جـــــــــــــامعة أبي بكـــر بلقـــايد – تـــلمســـــــــــــان –

Abou Bekr Belkaïd University – Tlemcen –
Faculty of TECHNOLOGY

Graduation Theses

Presented for obtaining the **MASTER'S degree**

In: Mechanical Engineering

Speciality: Mechanical Construction

By: TABET AOUL Lokmane

Title of the Thesis:

Study and Design an Injection Mould for Plastic Materials: Enhancing the Integration Rate of the Automotive Sector in Algeria

Publicly defended, on 16/07/2023, before the jury composed of:

M. BELALIA Sid Ahmed	**Pr**	University of Tlemcen	President
M. CHORFI Mohammed	**MCA**	University of Tlemcen	Examiner
M. SIFI Miloud	**MCA**	University of Tlemcen	Examiner
M. MAACHOU Omar	**MCA**	University of Tlemcen	Examiner
M. HAMZA CHERIF S. Mohammed	**MCA**	University of Tlemcen	Supervisor
M. SERDOUN Nadjib	**MCA**	ESSA of Tlemcen	Co-Supervisor

Year of study: 2022 /2023

Acknowledgments

We would like to express our heartfelt gratitude to all individuals and organisations who have contributed to the success of this project. Their support, guidance, and collaboration have been instrumental in bringing our vision to life. We would like to acknowledge the following:

The Team: *We extend our deepest appreciation to the dedicated members of our project team who have invested their time, knowledge, and skills in making this project a reality. We are grateful for the guidance and support of M. HAMZA CHERIF and M. SERDOUN Nadjib for their invaluable time and knowledge.*

FABIPLAST: *We would like to acknowledge FABIPLAST for their collaboration and partnership. We would like to extend a special thanks to M. Mansour for their expertise and guidance, which have greatly contributed to the success of this project.*

Members of the Jury: *We would like to extend our heartfelt appreciation to the esteemed members of the jury, M. BELAILA S. Ahmed, M. CHORFI Mohammed, M. SIFI Miloud and M. MAACHOU Omar, for generously dedicating their time, sharing their expertise, and providing invaluable insights during the evaluation of this thesis. Their constructive feedback and guidance have significantly contributed to the improvement and refinement of our work. We are truly grateful for their unwavering commitment to academic excellence.*

Friends and Family*: Lastly, we would also like to express our heartfelt gratitude to our friends and family for their unwavering support, understanding, and encouragement. Their belief in our abilities and their constant encouragement have been a source of motivation and strength.*

Dedication

We dedicate this project to all those who have been a source of inspiration, support, and encouragement throughout our journey. Their unwavering belief in our abilities and their constant encouragement have fuelled our determination to overcome challenges and achieve our goals. This dedication is a tribute to:

The Family*: To my parents, your unconditional love, understanding, and unwavering support have been the driving force behind my pursuit of excellence. I am deeply grateful for the encouragement, sacrifices, and assistance you have provided. This thesis is dedicated to you with heartfelt appreciation.*

The Mentors and Educators: *For their guidance, wisdom, and knowledge-sharing. Your expertise and mentorship have shaped our skills and transformed us into confident professionals.*

The Team: *For your dedication, hard work, and collaboration. Your commitment and teamwork have been instrumental in turning ideas into reality and overcoming obstacles along the way.*

The Supporters and Well-wishers: *For your belief in our vision and unwavering encouragement. Your words of encouragement and positive energy have motivated us to strive for greatness.*

The Partners and Collaborators: *For your collaboration, expertise, and shared vision. Your contributions and partnership have added value to our project and helped us achieve new heights*

خلاصة

يلعب قطاع السيارات دورًا حاسمًا في التنمية الاقتصادية للدول. في الجزائر، تشهد صناعة السيارات فرصًا للنمو، نتيجة الطلب المتزايد على منتجات وأجزاء بلاستيكية.

الهدف الرئيسي لهذا البحث هو دراسة وتصميم قالب حقن لقطعة بلاستيكية، مع التركيز بشكل خاص على غطاء مقبض باب السيارة، لتعزيز معدل الادماج في قطاع السيارات في الجزائر.

لقد استكشفنا نمذجة القطع ثلاثية الأبعاد، ومحاكاة الحقن، وتصميم قالب حقن لغطاء مقبض الباب الداخلي لطراز سوزوكي ساموراي.

ستوفر نتائج هذا البحث نقاط نظر قيمة لمصممي ومصنعي القوالب، بالإضافة إلى الآثار العملية لصناع القرار. بالإضافة إلى ذلك، ستعزز هذه البحوث فرص التعاون ونقل المعرفة والابتكار في تصميم القوالب لمكونات السيارات.

Abstract

The automotive sector has a crucial role in the economic development and integration of nations. In Algeria, the automotive industry is witnessing growth opportunities, driven by increasing demand for plastic products and parts.

The primary objective of this research is to study and design an injection mould for a plastic part, specifically focusing on a car door handle cover, to enhance the integration rate of the automotive sector in Algeria.

We have explored the 3D part modelling, injection simulation, and mould design to develop the interior door handle cover for the SUZUKI SAMURAI model.

The results of this research will provide valuable insights for mould designers and manufacturers, as well as practical implications for policymakers. Additionally, this research will foster collaboration opportunities, knowledge transfer, and innovation in mould design for automotive components.

Résumé

Le secteur automobile joue un rôle crucial dans le développement économique et l'intégration des nations. En Algérie, l'industrie automobile connaît des opportunités de croissance, stimulées par une demande croissante de produits et de pièces en plastique.
L'objectif principal de cette recherche est d'étudier et de concevoir un moule d'injection pour une pièce en plastique, en mettant spécifiquement l'accent sur un cache pour une poignée de porte de voiture, afin d'améliorer le taux d'intégration du secteur automobile en Algérie.

Nous avons exploré la modélisation en 3D de la pièce, la simulation d'injection et la conception du moule pour développer le cache de la poignée de porte intérieure du modèle SUZUKI SAMURAI.

Les résultats de cette recherche fourniront des informations précieuses pour les concepteurs et fabricants de moules, ainsi que des implications pratiques pour les décideurs. De plus, cette recherche favorisera les opportunités de collaboration, le transfert de connaissances et l'innovation dans la conception de moules pour les composants automobiles.

Table of contents

LIST OF FIGURES

LIST OF FIGURES

LIST OF TABLES

LIST OF ACRONYMS

Word	Abbreviation
SUZUKI SAMURAI	**SS**
Sports Utility Vehicle	**SUV**
SolidWorks	**SW**
Computer Aided Design	**CAD**
Finite element analysis	**FEA**
Acrylonitrile Butadiene Styrene	**ABS**
Polypropylene	**PP**
Polystyrene	**PS**
Polyethylene	**PE**
Polycarbonate	**PC**
Polyvinyl Chloride	**PVC**

GENERAL INTRODUCTION

GENERAL INTRODUCTION

Introduction

The injection moulding process is a fundamental manufacturing technique used in various industries, including the automotive sector. It enables the production of complex and precise parts with high efficiency and accuracy. This work explores the injection moulding process and its application in the design and manufacturing of the interior door handle of the SUZUKI SAMURAI car.

Through the exploration of the injection moulding process, part design, part simulation, and mould design, this work aims to contribute to the advancement of the automotive sector. By developing innovative and optimized interior car door handle cover designs and efficient manufacturing processes, we strive to enhance the integration rate, promote local production, and foster economic growth. The findings of this work provide valuable insights for designers, manufacturers, and policymakers in the automotive industry, paving the way for further advancements and opportunities.

We have covered four chapters, and this include:

Chapter I provides an overview of the injection moulding process, including the injection cycle, injectables, injection machines, and injection moulds. It sets the foundation for understanding the key elements and stages involved in the process. The chapter concludes with a summary of the importance and relevance of the injection moulding process in the manufacturing industry.

Chapter II focuses on part design, which plays a crucial role in achieving functional and aesthetically pleasing interior car door handle covers. It explores design rules, definition drawings, part specifications, and the overall design process. The chapter also discusses the importance of design verification in ensuring the quality and performance of the designed parts.

Chapter III delves into part simulation; a powerful tool used to analyse and optimize the interior car door handle cover design. It introduces the concept of simulation analysis and discusses its application in evaluating stress distribution and predicting part performance. The chapter concludes by highlighting the significance of simulation in enhancing the design and manufacturing process.

Chapter IV shifts the focus to mould design, an essential aspect of the injection moulding process. It introduces mould rules and functions, emphasizing the importance of meticulous core and cavity design. The chapter also presents detailed assembly and detail drawings of the mould components. It concludes with an overview of the role of mould design in ensuring high-quality and efficient production.

CHAPTER I
Introduction to Injection Moulding Process

1.1 Overview of the Injection Moulding Process

Injection moulding is a widely used manufacturing process in which molten plastic is injected into a mould cavity to form a desired shape. This process allows for the rapid production (single operation) of high-quality objects in large quantities, even if the shapes are complex, in weight ranges ranging from a few grams to several kilograms. Plastic injection is used in a wide range of fields: automotive, toys, electronics, robotics, aerospace, medical...

This section provides an overview of the injection moulding process, including the injection cycle, injectables, injection machines, and injection moulds **[1]**.

1.2 Injection Cycle

The injection cycle refers to the sequence of steps involved in the injection moulding process. It typically consists of several stages, including material preparation, injection, cooling, and ejection. These steps ensure the proper execution of the injection moulding process. The key steps involved in the injection cycle are as follows:

- **Closing and Locking the Mould:**

At the beginning of the cycle, the mould is closed and securely locked (see Figure I.1). This ensures that the two halves of the mould, namely the core and cavity, are aligned and held in place during the injection process.

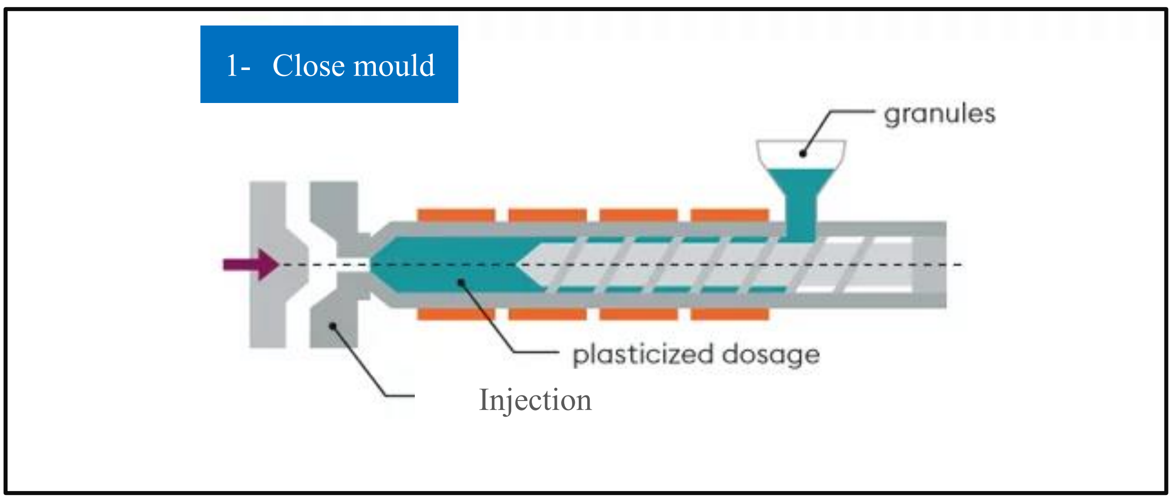


Fig.I.1: closing and locking the mould [2]

- **Injection of the Molten Material and Maintenance under Pressure:**

Once the desired amount of molten material has been accumulated, it is injected into the mould cavity under high pressure. The injection phase involves the rapid movement of the plasticizing group to push the molten material into the mould. After injection, the pressure is maintained for a specified period to ensure proper filling and packing of the mould cavity.

Figure I.2 represent the process of injection of the molten material and maintenance it under pressure:

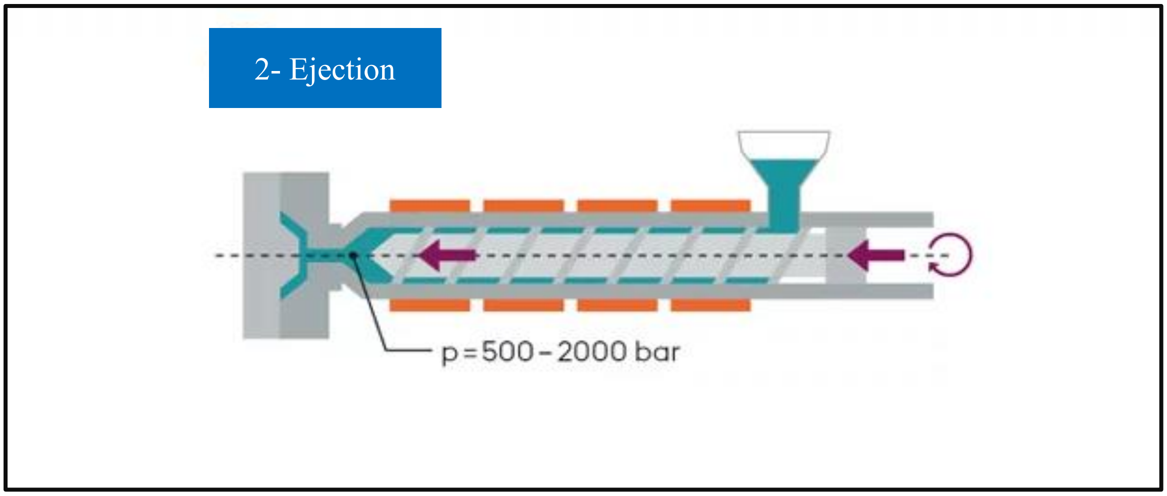


Fig.I.2: injection of the molten material and maintenance under pressure [2]

- **Pressing and Cooling:**

After the injection phase, the moulded part is left to cool and solidify within the mould while maintaining the pressure. The cooling process allows the molten plastic to take the desired shape and achieve the required dimensional stability.

Figure I.3 represent the pressing and cooling process:

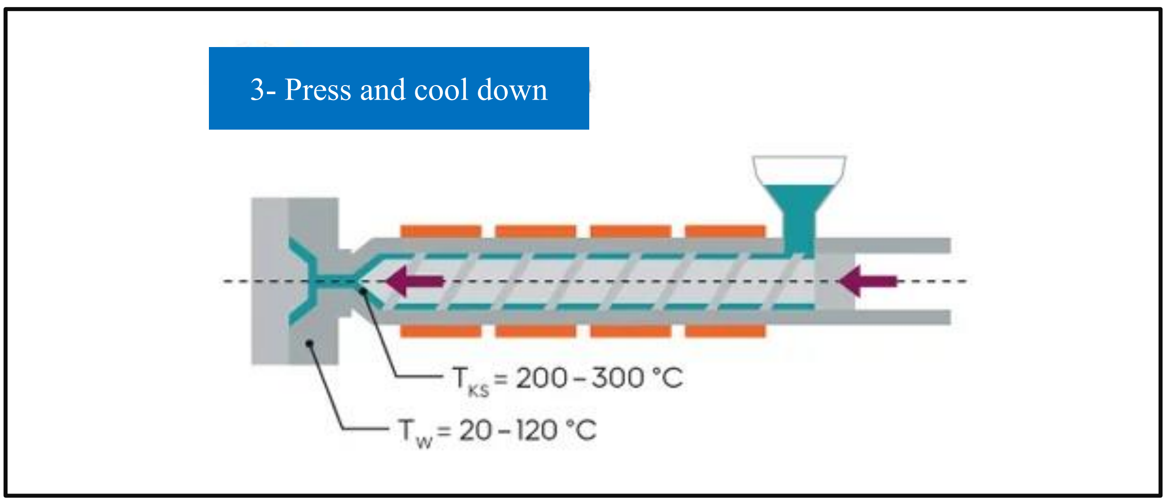


Fig.I.3: pressing and cooling [2]

- **Plasticizing and Dosage:**

The plasticizing group, comprising the reciprocating screw or plunger, advances to move the plastic material from the hopper into the heating barrel. This step facilitates the gradual melting and homogenization of the plastic material.

During this step, a predetermined volume of molten plastic material is fused and accumulated in the heating barrel. The lamination process ensures the proper fusion of the material and prepares it for the subsequent injection.

Figure I.4 represent the plasticizing and dosage process:

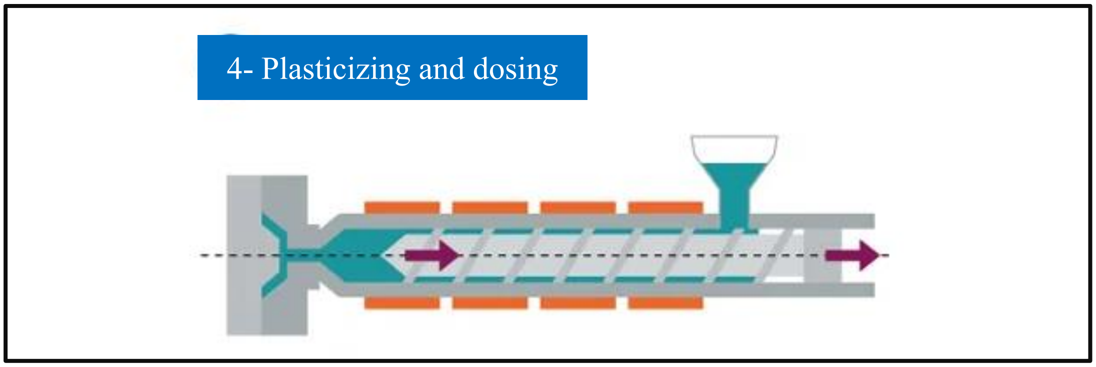


Fig.I.4: plasticizing and dosage [2]

- **Mould Opening and Parts Ejection:**

Once the moulded part has sufficiently cooled and solidified, the mould is opened. This step involves separating the two halves of the mould, namely the core and cavity, to reveal the moulded part.

After opening the mould, the moulded part is ejected from the mould cavity. Ejection can be accomplished using various mechanisms such as ejector pins or robotic arms. Once the part is removed, the mould is ready for the next cycle, starting from the closing and locking step.

Figure I.5 represent the mould opening and ejection process:

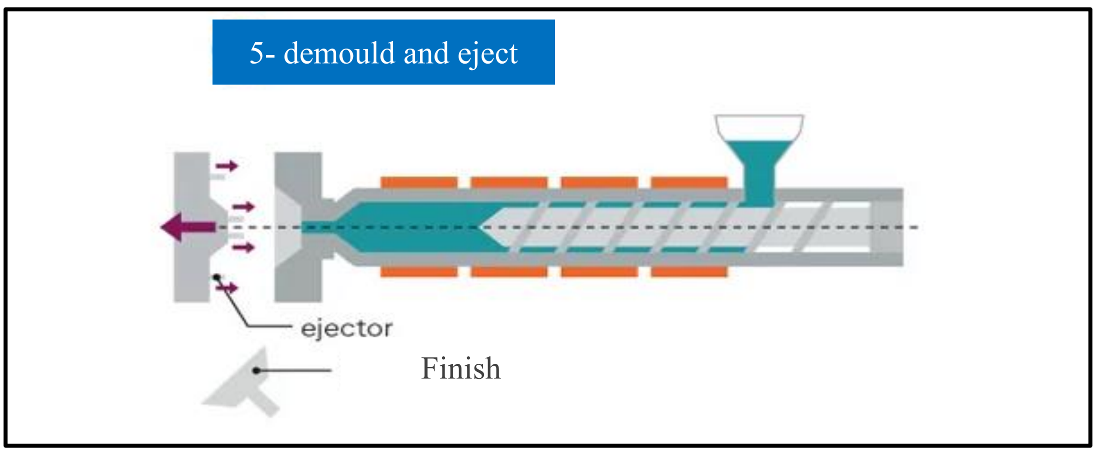


Fig.I.5: mould opening and parts ejection [2]

1.3 Injectables

Injectables are the materials used in the injection moulding process. Thermoplastics, such as PP, PE, and PS, are commonly used due to their excellent mouldability and versatility. Thermosetting plastics, such as epoxy and phenolic resins, are also utilized in specific applications that require superior heat resistance and dimensional stability **[3]**.

Table 01 is an example the list the injectable materials commonly used in the injection moulding process along with some specifications:

Table 01: commonly used injectable materials [3]

Injectable Material	Properties	Typical Applications
PP	- High impact strength - Good chemical resistance - Low density	- Automotive components - Packaging - Consumer products
PS	- Good dimensional stability - Excellent surface finish - Low moisture absorption	- Electronics - Food packaging - Medical devices
ABS	- High heat resistance - Excellent impact resistance - Good surface finish - Chemical resistance	- Automotive components - Electrical enclosures - Appliances
PE	- Good flexibility - Excellent chemical resistance - Low friction coefficient	- Bottles and containers - Agricultural products - Toys and sporting goods
PC	- High transparency - Excellent impact resistance - Good electrical insulation	- Optical components - Automotive lighting - Safety helmets
PVC	- Flame retardant - Good weather resistance - Chemical stability	- Building and construction - Vinyl flooring - Medical tubing

1.4 Injection Machines

Injection machines, also known as injection moulding machines, are used to perform the injection process. These machines consist of a hopper, where the plastic material is fed, and a heating barrel, where the material is melted. The molten plastic is then injected into the mould cavity using a reciprocating screw (injection ram). Injection machines vary in size and capacity depending on the requirements of the moulded part.

- **The Injection Unit:**

The injection unit (see Figure I.6) plays a critical role in the injection moulding process by heating and injecting polymer material into the mould. A key component of this unit is the hopper, designed to hold a significant quantity of polymer pellets. To initiate the operation, the machine is fed with raw material, which undergoes a heating process until the polymer reaches a liquefied state. The liquefied polymer is then injected into the mould.

To facilitate the movement of the material, a filtered shaft known as a screw is employed. The screw features varying diameters along its length, effectively increasing the force applied to push the material up to the injection threshold. The cooling of this mould component is typically achieved through the use of water. The screw of the machine operates in two modes, rotation and advancement (or axial translation), to generate high-pressure thrust on the plastic material. This process is facilitated by a hydraulic circuit. As a result of the friction involved in the injection process, an additional form of energy is generated, known as a "source of dissipation."

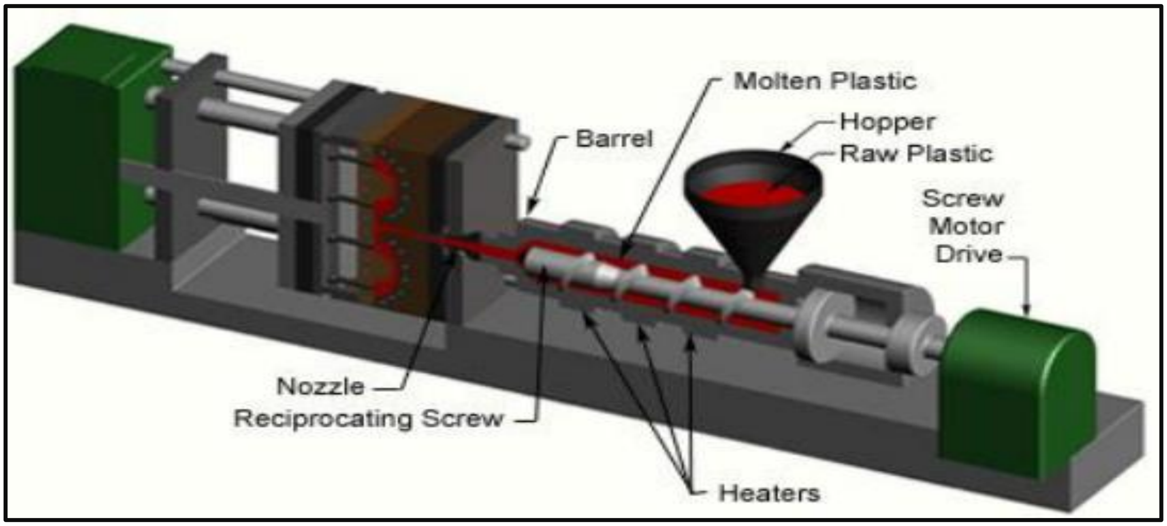


Fig.I.6: the injection unit [4]

- **The Clamping Unit**:

The clamping unit is a vital component of the injection moulding machine responsible for securely holding the mould in place during the injection process. It ensures that the mould remains closed with sufficient force to withstand the high pressure exerted by the injected material.

The clamping unit (see Figure I.7) consists of two primary components: the mould and the clamping mechanism. The mould is designed to create the desired shape and structure of the final product. It is typically made of durable materials such as steel and is precision-engineered to withstand the forces and temperatures involved in the injection moulding process.

The clamping mechanism is responsible for exerting the necessary force to keep the mould closed. It typically comprises hydraulic or mechanical systems that apply pressure to the mould. Hydraulic clamping units utilize hydraulic cylinders and pistons to generate and control the clamping force, while mechanical clamping units employ mechanical mechanisms like toggle systems.

The clamping force required depends on factors such as the size and complexity of the mould, the material being injected, and the desired quality of the final product. It is crucial to apply the appropriate clamping force to ensure proper mould closure and prevent any flash or distortion in the moulded parts.

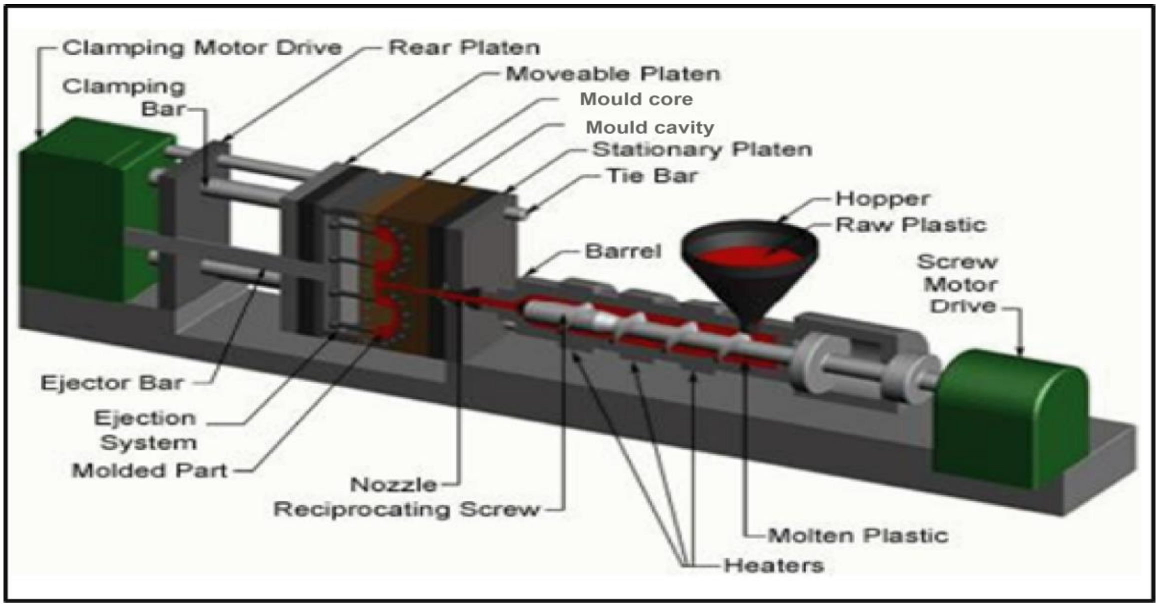


Fig.I.7: the clamping unit [4]

1.5 Injection Moulds

Injection moulds, also known as dies or tools, play a critical role in the injection moulding process by enabling the production of precise and customized plastic parts. These moulds are meticulously designed and manufactured to meet specific requirements and are typically crafted from high-grade steel for durability and longevity.

An injection mould consists of two essential parts: the core and the cavity. The core and cavity are meticulously shaped components that define the final design of the plastic part being produced. The core creates the internal features of the part, while the cavity shapes the external features. These components work together to form the desired shape and geometry of the plastic part.

The core and cavity (see Figure I.8) are designed with great precision to ensure accurate reproduction of the intended shape and dimensions. The surfaces of the core and cavity are carefully finished to achieve the desired surface finish of the final part. This level of precision ensures that each produced part meets the required specifications and quality standards **[5].**

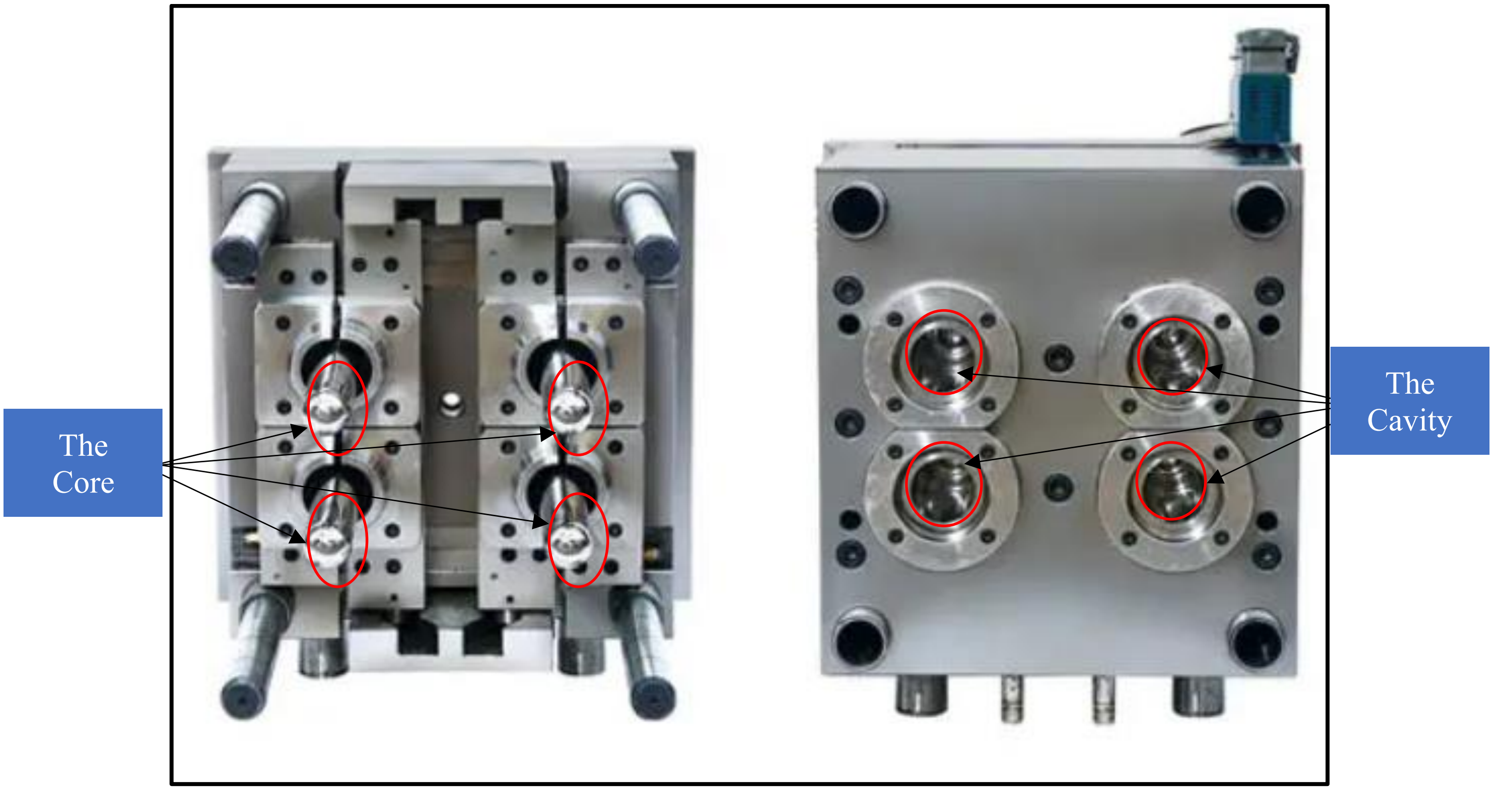


Fig.I.8: the core and cavity [5]

In addition to the core and cavity components, injection moulds incorporate several other essential elements that contribute to the overall functionality and efficiency of the moulding process. These components work together to ensure proper ejection of the finished part from the mould. Some of these components include ejector bars, pins, plates, locating rings, sprue bushings, and more.

- **Ejector bars:**

Ejector bars are used to push or eject the finished part from the mould once the cooling and solidification process is complete. They are typically positioned on the back side of the mould and are actuated by an ejector system. The ejector bars apply force to the moulded part, allowing it to be released smoothly.

- **Ejector pins:**

Ejector pins, also known as knockout pins, are slender rods or pins that are used to push the part out of the mould. They are strategically positioned within the mould and are actuated by the ejector system. Ejector pins are designed to withstand the forces involved in the ejection process and ensure the proper release of the part.

- **Ejector plates:**

Ejector plates are large plates or mechanisms that house the ejector pins. They provide the necessary support and stability to the pins during the ejection process. Ejector plates are often operated by hydraulic or mechanical systems and play a crucial role in ensuring efficient part ejection.

- **Locating rings:**

Locating rings are precision-machined components that aid in the accurate alignment and assembly of the mould. They are typically installed on the parting line of the mould and help maintain proper alignment between the core and cavity components during mould closure. Locating rings ensure precise registration of the mould halves, resulting in consistent part quality.

- **Sprue bushing:**

The sprue bushing is a component located at the entry point of the mould, where the molten plastic material is injected. It provides a channel for the plastic material to flow into the mould cavity. The sprue bushing is designed to withstand the high pressure and temperature of the molten plastic material during injection.

These components (see Figure I.9), along with the core and cavity, work harmoniously to achieve successful injection moulding. Their proper design, positioning, and functionality are crucial for achieving consistent part quality, efficient ejection, and overall mould performance.

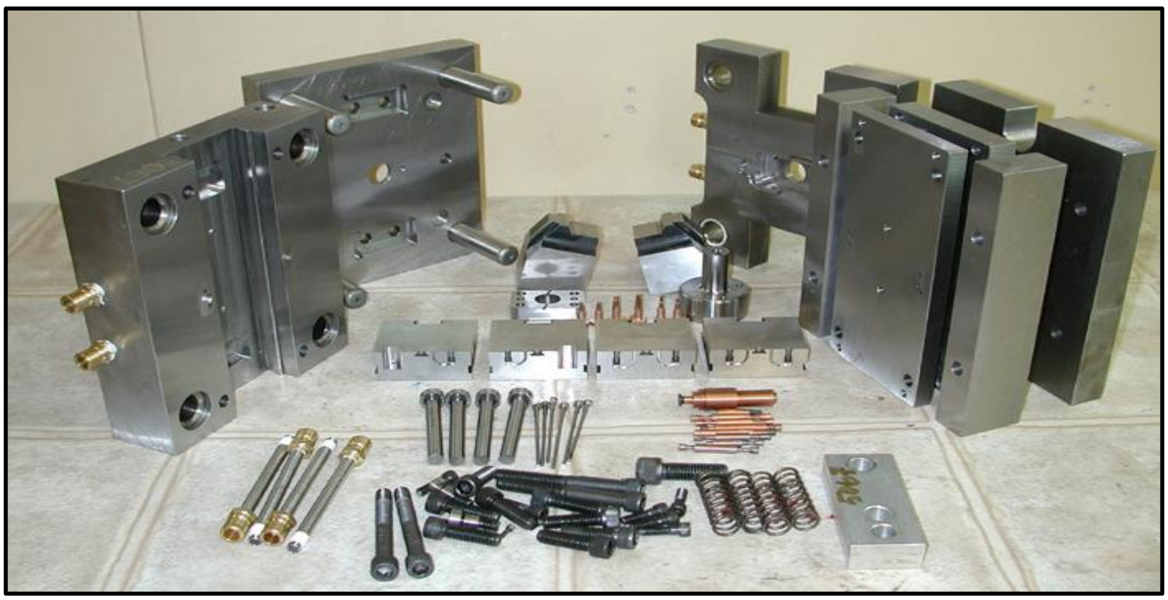


Fig.I.9: mould components [6]

Another critical aspect of injection mould design is the incorporation of cooling channels. Cooling channels are strategically placed within the mould to facilitate the efficient and uniform cooling of the molten plastic (Figure I.10). The cooling process is crucial in maintaining the structural integrity and dimensional accuracy of the part. The cooling channels allow the heat to dissipate, reducing the cycle time and ensuring proper solidification of the plastic material.

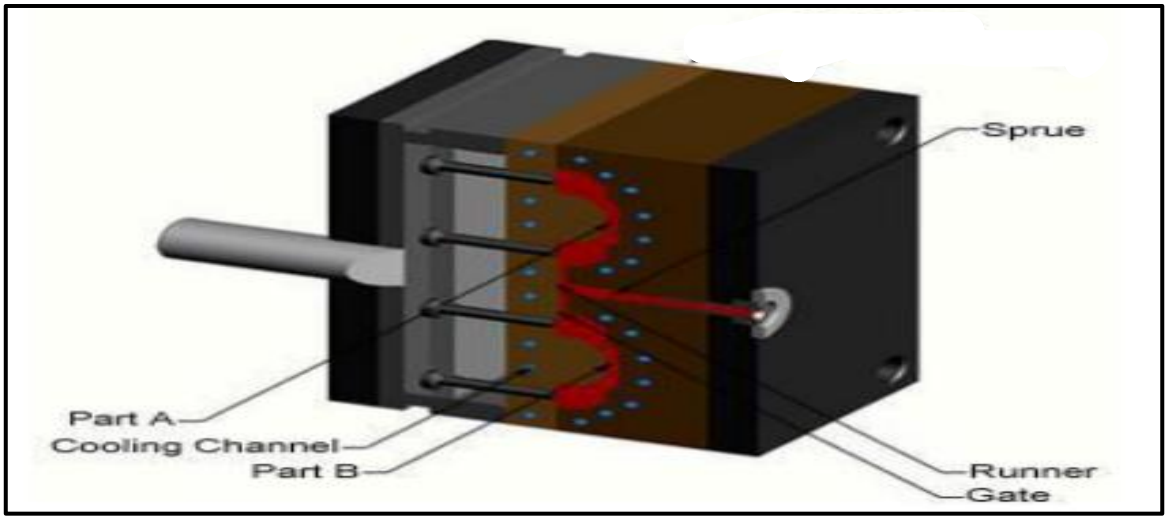


Fig.I.10: cooling channels [4]

Injection moulds, with their intricately designed core and cavity components, all the essential elements and the efficient cooling channels, are instrumental in achieving high-quality, dimensionally accurate plastic parts. The precision engineering and attention to detail in mould design contribute to the overall success and efficiency of the injection moulding process.

1.6 Conclusion

In conclusion, this chapter has provided an introduction to the injection moulding process, offering an overview of its various components and stages. We began by highlighting the importance of injection moulding as a widely used manufacturing method for producing plastic parts. We discussed the key elements involved in the injection cycle, including the heating and injection of the polymer material, as well as the cooling and ejection of the finished part.

Furthermore, we explored the concept of injectables, emphasizing the diverse range of materials that can be used in the injection moulding process. Additionally, we touched upon the significance of injection machines, which are responsible for executing the injection process with precision and efficiency.

CHAPTER II: Part Design

2.1 Introduction

In the field of product design, the creation of functional and aesthetically pleasing parts is a crucial aspect of developing innovative and user-centric products. Part design plays a pivotal role in shaping the overall appearance, functionality, and user experience of a product. In this chapter, we will delve into the fascinating world of part design, with a specific focus on the Suzuki Samurai interior door handle cover.

The Suzuki Samurai represented in Figure II.1, a popular compact SUV, is renowned for its reliable performance and rugged design. As we explore the design process of its interior door handle, we will uncover the principles, considerations, and techniques involved in creating a successful part. The interior door handle serves as an essential component, providing convenient access and enhanced ergonomics for the vehicle occupants.

Fig.II.1: SUZUKI SAMURAI interior door handle [11]

Using the powerful software SW 2022 **[9]**, we have designed a door handle that not only meets the functional requirements but also harmonizes with the overall aesthetics of the SS. Throughout the design process, we employed various features and tools, such as Extruded Boss/Base, Extruded Cut, Fillet, and Shell, to meticulously craft the intricate details of the door handle.

The chosen part, the SS interior door handle, exemplifies the complex nature of part design, where form and function intersect. We aimed to create a design that not only embodies the brand's identity but also prioritizes user comfort, ease of use, and durability.

2.2 Design rules

Designing the Suzuki Samurai interior door handle cover requires adherence to specific design rules to ensure optimal functionality and manufacturability. These design rules encompass crucial aspects, here are some important rules to consider:

- **Parting Line:**

The parting line is a critical aspect of part design in injection moulding. It defines the boundary between the two halves of the mould, allowing for the separation of the part after it is moulded. The location and design of the parting line significantly impact the quality and manufacturability of the final product. Here are the key considerations for parting line design:

- **Determining the Parting Line:** Analysing the part geometry and complexity helps in identifying the most suitable location for the parting line. It should align with the natural flow of the molten plastic and minimize the need for complex Mold features.
- **Simplifying Parting Line Geometry:** Designing a straight, linear parting line is often preferred as it simplifies mould design and reduces costs. However, in cases where the part geometry demands a non-linear parting line, careful attention should be given to maintaining consistent wall thickness and avoiding sharp angles that could lead to cosmetic defects or mould damage.
- **Minimizing Parting Line Visibility:** Depending on the desired aesthetics of the part, efforts should be made to position the parting line in less visible areas or incorporate design features to conceal it. This can enhance the overall appearance of the finished product.

- **Moulding Direction:**

The choice of moulding direction plays a vital role in achieving high-quality parts with optimal structural integrity. The moulding direction determines how the molten plastic flows and fills the mould cavity. Consider the following aspects when determining the moulding direction:

- **Flow Path Length:** Select a moulding direction that minimizes the flow path length, as longer flow paths can lead to increased cycle time and potential defects such as flow marks or air traps.

- **Avoiding Undercuts:**

Ensure that the chosen moulding direction does not result in undercuts, which are features that prevent the part from being easily ejected from the mould. Undercuts can complicate the mould design and increase production costs.

- **Wall Thickness Consistency:**

Maintain uniform wall thickness throughout the part by considering the moulding direction. This helps in achieving balanced cooling and preventing sink marks or warpage.

- **Draft Angle:**

Draft angles are essential design elements that facilitate the smooth ejection of parts from the mould. They are included on vertical surfaces to allow easy release without causing damage. Consider the following guidelines for draft angle design:

- **Determining Draft Angle Values:** The recommended draft angle varies based on the material, surface finish, and part geometry. Generally, draft angles between 1° and 3° are suitable for most applications, but more complex geometries may require larger draft angles.

- **Avoiding Undercuts:** Incorporate draft angles to prevent undercuts, which can hinder part ejection and cause mould damage. Undercuts can occur at the intersections of walls, ribs, and other features.
- **Designing with Draft Analysis Tools:** Utilize draft analysis tools available in software such as SW 2022 **[9]** to evaluate and optimize draft angles. These tools provide valuable insights into potential moulding issues and help refine the design for better manufacturability.

- **Wall Thickness Guidelines:**

Maintaining appropriate wall thickness is crucial for achieving the desired part performance, structural integrity, and manufacturability. Here are the key considerations for wall thickness design:

- **Material Selection:** Different materials have specific recommended wall thickness ranges. Consult material datasheets or industry standards to determine the suitable wall thickness for the chosen material **[5]**.

- **Flow and Cooling Considerations:** Achieve uniform wall thickness to promote even material flow during injection moulding and ensure consistent cooling rates. Non-uniform wall thickness can result in defects such as sink marks or warpage **[5]**.

2.3 Definition drawings

The drawing in Figure II.2 provide detailed views, dimensions, tolerances, and material specifications of the interior car door handle cover. They serve as a reference for tooling design, mould making, and quality control during production.

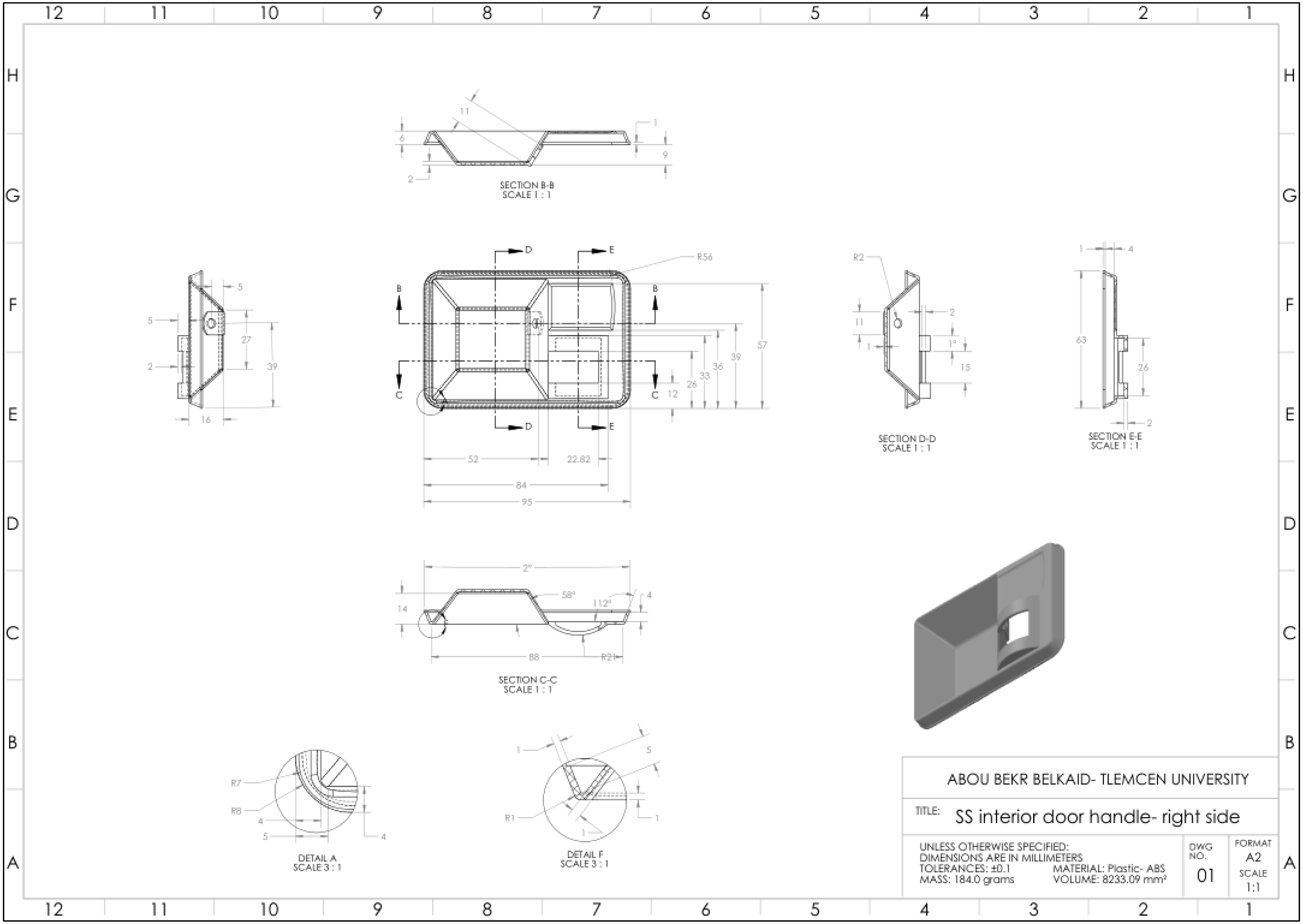


Fig.II.2: SS door handle- 2D drawing

2.4 Part specifications

When designing the interior car door handle cover for injection moulding, several specifications need to be considered. These specifications include the material used, volume, surface area, weight, and thickness of the part.

The Suzuki Samurai interior door handle has specific part specifications that must be taken into account during the design process **[7]**. These specifications include:

- **Material Selection:**

For the interior car door handle cover, ABS is a commonly used material due to its excellent mechanical properties, good impact resistance, and dimensional stability. ABS is known for its durability, heat resistance, and ability to be easily processed in injection moulding. The selection of ABS ensures that the door handle can withstand the demanding requirements of automotive applications. Table 02 showcase some important information about the ABS:

Table 02: material properties [9]

Property	Value	Units
Elastic Modulus	2000	N/mm²
Poisson's Ratio	0.394	N/A
Shear Modulus	318.9	N/mm²
Mass Density	1020	Kg/m³
Tensile Strength	30	N/mm²
Compressive Strength	/	N/mm²
Yield Strength	/	N/mm²
Thermal Expansion Coefficient	/	/K
Thermal Conductivity	0.2256	W/ (m.K)
Specific Heat	1386	J/(Kg.K)
Material Damping Ratio	/	N/A

- **Volume, Surface Area, Weight, and Thickness:**

The volume specification refers to the amount of space the interior car door handle cover occupies. It helps estimate the material requirements and manufacturing costs. The surface area specification refers to the total external area of the part, which affects the cooling time during the moulding process.

The weight specification provides an estimation of the material consumption and is important for cost calculations. The thickness specification refers to the wall thickness of the door handle. It is crucial to maintain uniform thickness throughout the part to ensure structural integrity, minimize warping, and prevent sink marks. Generally, the thickness of the door handle may vary depending on different sections, with thicker areas providing additional strength where needed **[9]**.

For the interior car door handle cover made of ABS, the specifications include a volume of approximately 9 (cm^3), a surface area of could range from 200 to 600 (cm^2), a weight of approximately 200 (g), and a recommended uniform thickness of 1.5 to 3 millimetres (mm) to ensure the part's functionality and durability **[12]**.

By considering these specifications, the design can be optimized for the desired performance, manufacturability, and cost-effectiveness. The melt temperature is 230 (C^o), the mould temperature is 80 (C^o) and the injection temperature is 90 (C^o) **[12]**.

2.5 Design Process

In the design phase of the SS interior door handle using SW 2022 **[9]**, various features and tools were employed to create a detailed and functional part. The software's powerful capabilities allowed for precise modelling and manipulation of the design elements.

This phase is a critical step in the development of a product, where the initial concept takes shape and transforms into a tangible design. In this process, various features and techniques are employed to create a functional and aesthetically pleasing part. This section provides a step-by-step guide to the part design process, highlighting the key features used.

We began by creating a sketch on the top plane, where we carefully defined the basic shape and dimensions of the part using various sketching tools. Next, we generated a new plan perpendicular to the top plane, allowing us to add more intricate features and complex geometry. With a new sketch on this plan, we moved on to the loft feature, which seamlessly connected the sketches from both planes, creating a smooth transition and achieving the desired design intent.

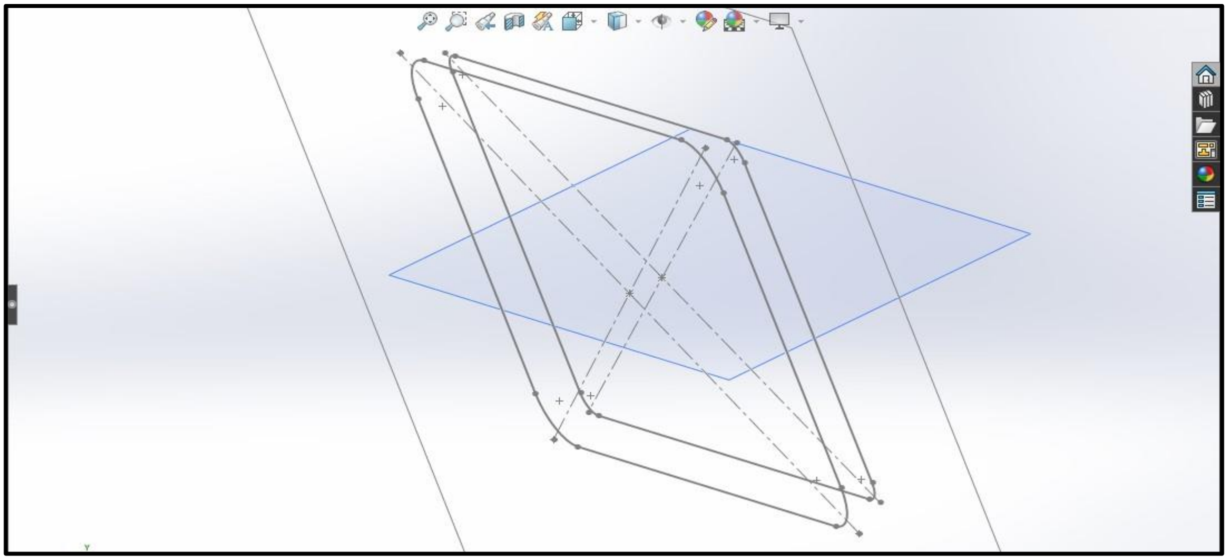


Fig.II.3: first step

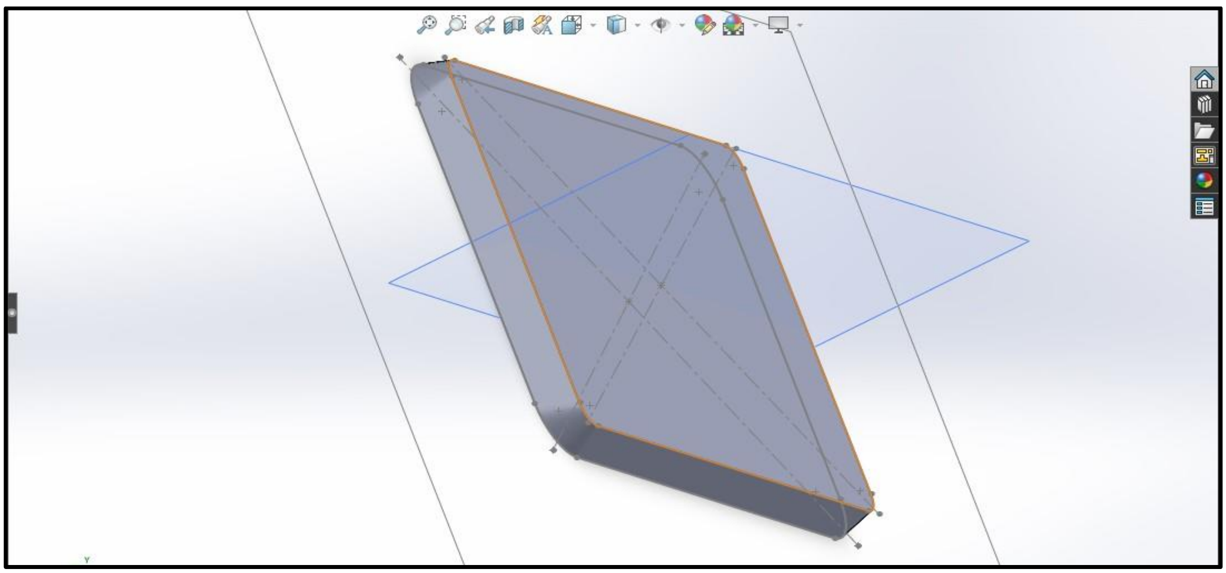


Fig.II.4: loft feature

To add thickness to the part, we employed the shell feature, specifying the desired wall thickness and allowing the software to remove material from the inside while maintaining the exterior geometry. This was particularly useful in creating hollow sections and optimizing the weight and material usage of the part. Using the extruded boss feature, we added material by extruding a sketch in a specific direction, shaping the part with raised features and protrusions. Conversely, the extruded cut feature enabled me to remove material, creating cavities or voids in the design.

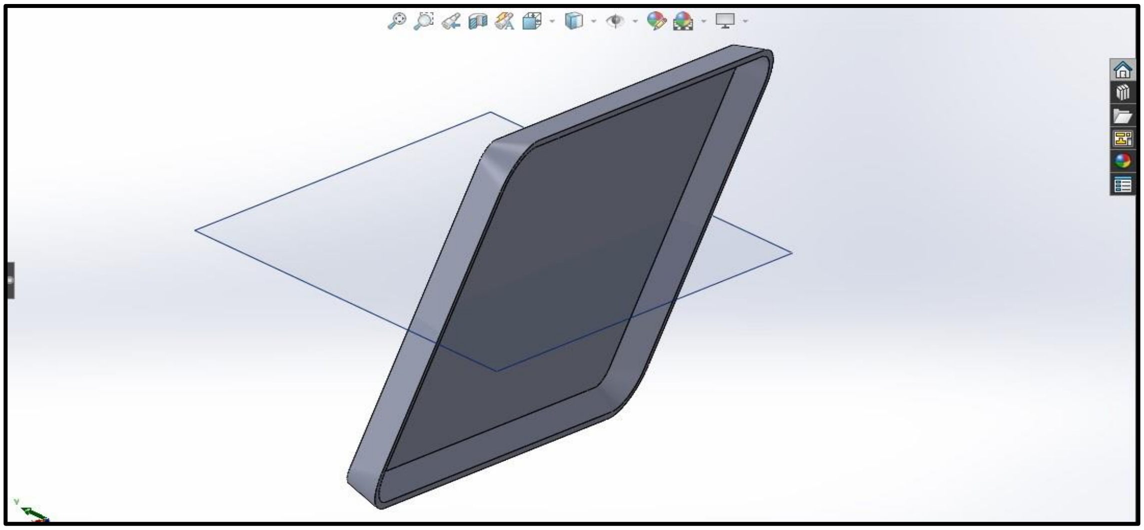


Fig.II.5: Shell feature

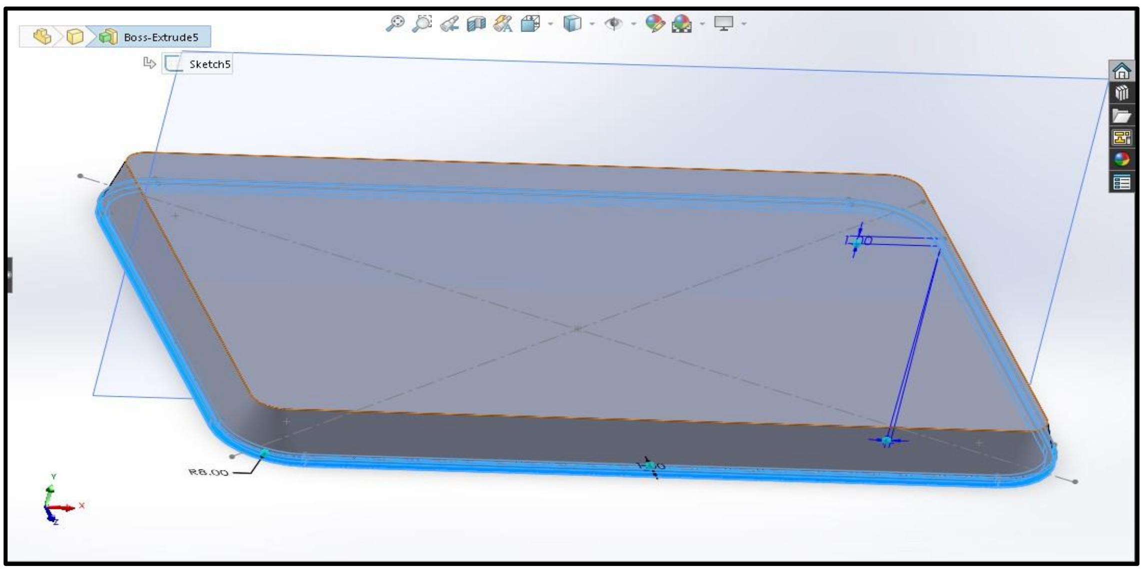


Fig.II.6: extruded boss feature

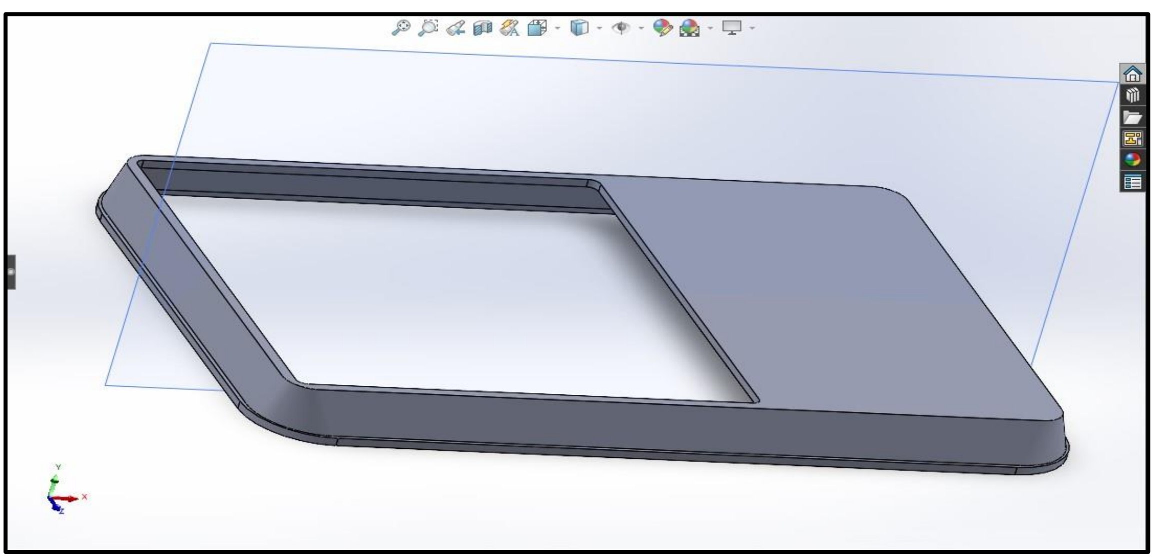


Fig.II.7: extruded cut feature

To refine the edges and enhance the aesthetics and durability of the part, we utilized the fillet feature. By specifying the desired fillet radius, the software automatically added rounded edges or fillets to the sharp corners, improving the overall appearance and reducing stress concentration. Throughout the process, we made sure to incorporate finishing touches as needed, using additional features such as extrude boss, extrude cut, shell, or loft, to add intricate details, refine surface finishes, and incorporate specific functional elements.

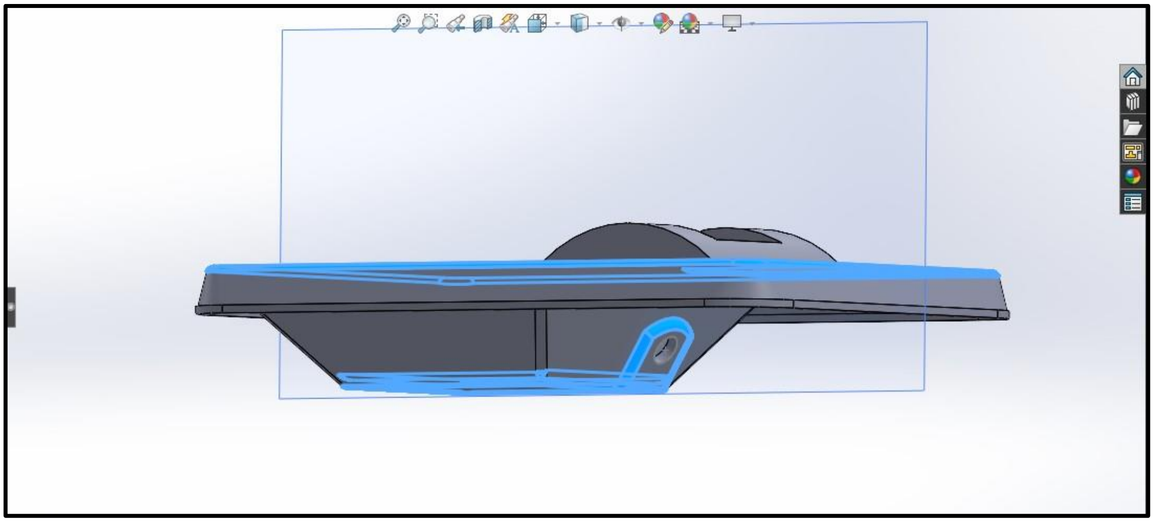


Fig.II.8: fillet feature

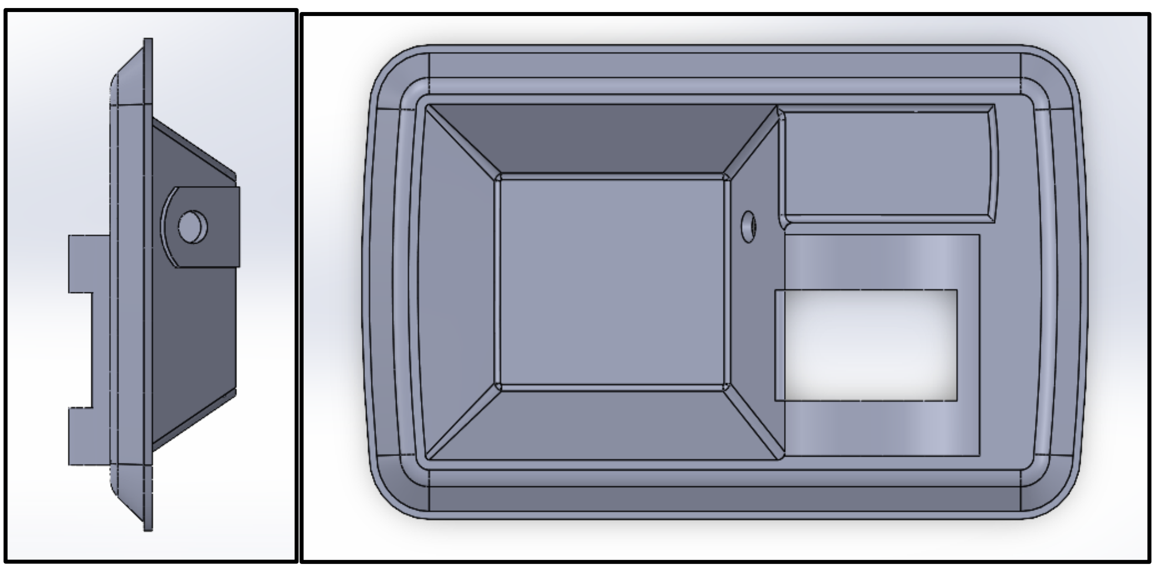


Fig.II.9: final design -top and right view

By utilizing the robust features of SW 2022 **[9]**, the design of the SS interior door handle was effectively translated into a digital model, ready for further verification and analysis.

2.6 Design Verification

The static simulation conducted in this study aimed to analyse the behaviour of the part when the handle is fixed onto the door. By examining the stress distribution and displacements, we sought to gain a deeper understanding of how the part responds to the applied loading conditions. This simulation mimicked the real-world scenario of attaching the handle to the door, enabling us to assess its structural integrity and performance.

- **Simulation Steps:**
  - **Fixing the Core on Face 1:** In order to accurately simulate the handle's attachment to the door, we fixed the core of the handle onto Face 1 of the door. This fixation was achieved using reference geometry, ensuring a secure and rigid connection between the handle and the door during the analysis.

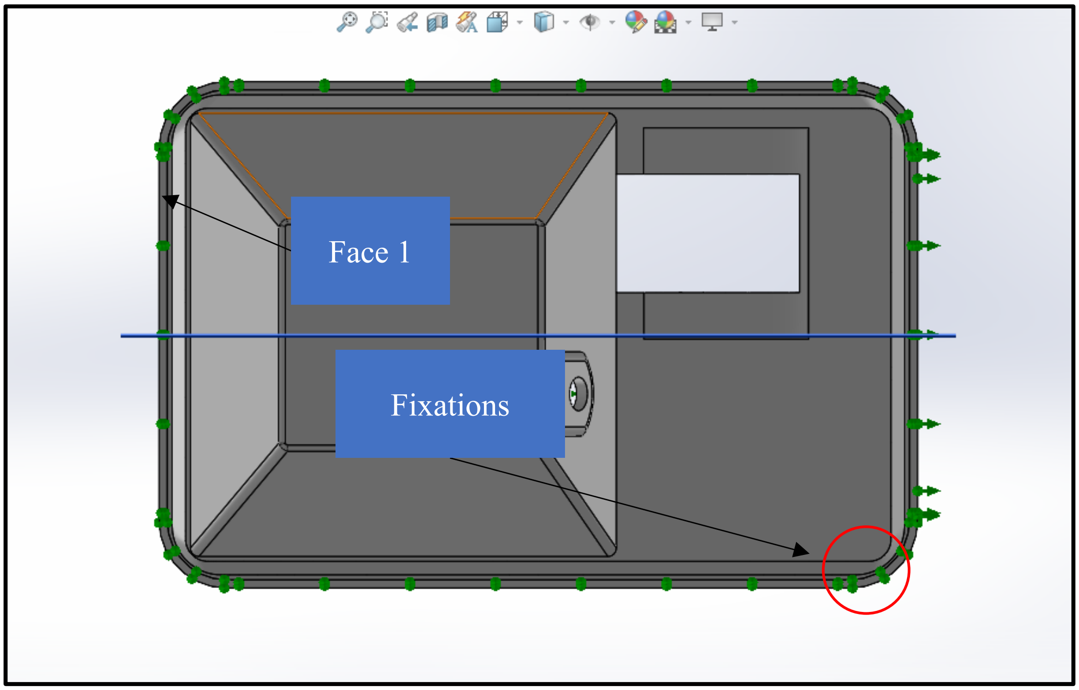


Fig.II.10: Fixing the Core on Face 1

  - **Applying Displacement on Face 2:** To simulate the pressure exerted by the screw when fixing the handle, a displacement of 1mm was applied to Face 2 of the door. This displacement represented the mechanical force exerted during the fastening process, allowing us to evaluate the resulting stress distribution and deformation patterns within the part.

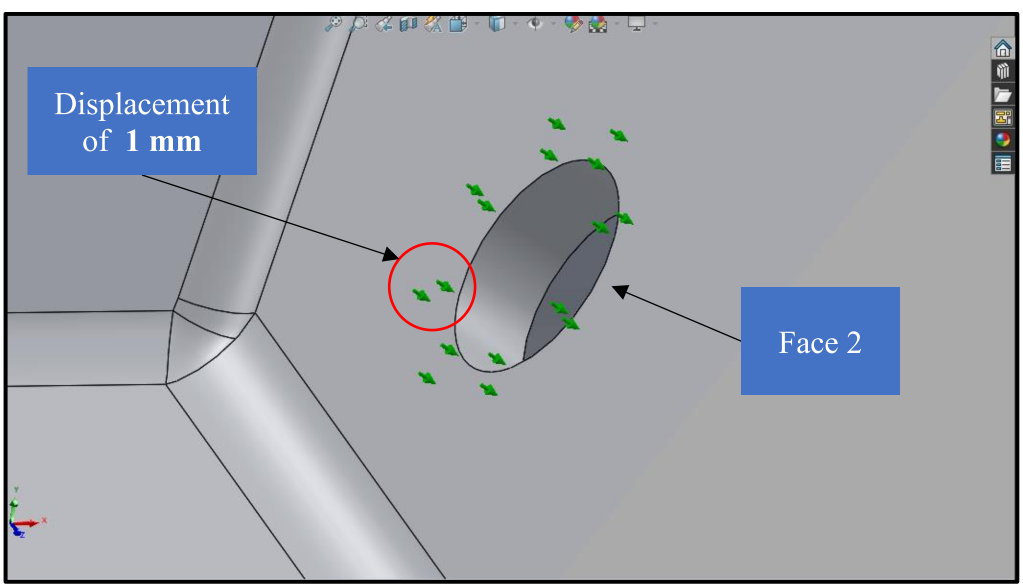


Fig.II.11: Applying Displacement on Face 2

The next step would be to create the mesh, and run the analysis.

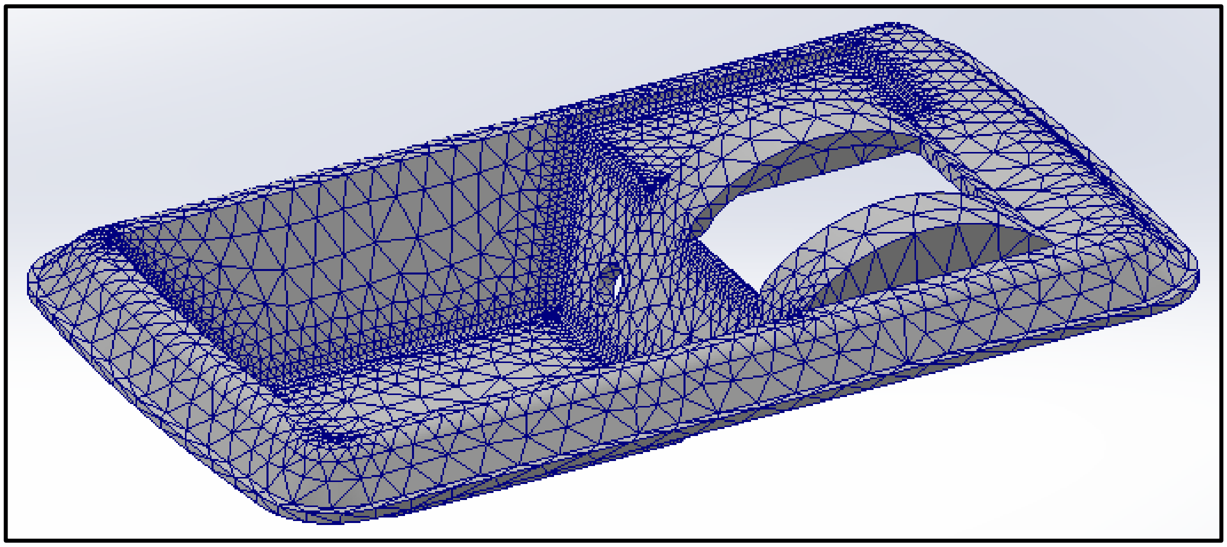


Fig.II.12: mesh created

By following these simulation steps, we were able to get the results bellow, noting that the findings from this simulation will guide us in identifying areas of concern and making informed design improvements to enhance the handle's reliability and durability in real-world applications.

- **Plastic Structure Test:** Plastic structure tests, such as stress analysis (von Mises) and displacement analysis (Residual displacement) are important aspects of evaluating the structural integrity and performance of plastic components. These tests help to understand how the material responds to various loads and deformations, ensuring that the part can withstand the expected operating conditions.

- **Stress analysis:** represented by the von Mises stress, measures the combined effect of all stress components within the material. It provides insights into areas of high stress concentration, which can help identify potential failure points or regions that require design modifications to ensure structural integrity.

- **Displacement analysis:** represented by residual displacement, examines the deformations and displacements that occur in the part after the application of loads. It helps to understand how the part will behave under different conditions and can be used to assess the suitability of the design for its intended application.

- **Interpretation of Results:**

The results obtained from the static simulation provide valuable information about the stress distribution and displacements within the part when the handle is fixed onto the door. Here is an interpretation of the findings based on the provided values:

- **Stress Analysis:** The von Mises stress is a measure of the overall stress experienced by the part, considering both the magnitude and distribution of stresses. Please refer to Figure II.13 for a visual representation of the stress distribution.

The minimum von Mises stress recorded was 0.03933 N/mm², represented by the dark blue colour in the figure. This indicates areas of relatively lower stress within the part. On the other hand, the maximum von Mises stress was 3.437 N/mm². These regions signify areas of high stress concentration within the part.

It is important to note that the tensile strength of the ABS plastic used in the analysis is 30 N/mm². This provides a reference point for evaluating the stress levels. Comparing the maximum von Mises stress of 3.437 N/mm² with the material's tensile strength, it is evident that the part experiences stresses well below its strength capacity. This indicates that the part should be able to withstand the applied loading conditions without significant risk of failure or deformation.
Based on this analysis, we can conclude that the plastic part exhibits a stress distribution that is within the acceptable range of the material's tensile strength.

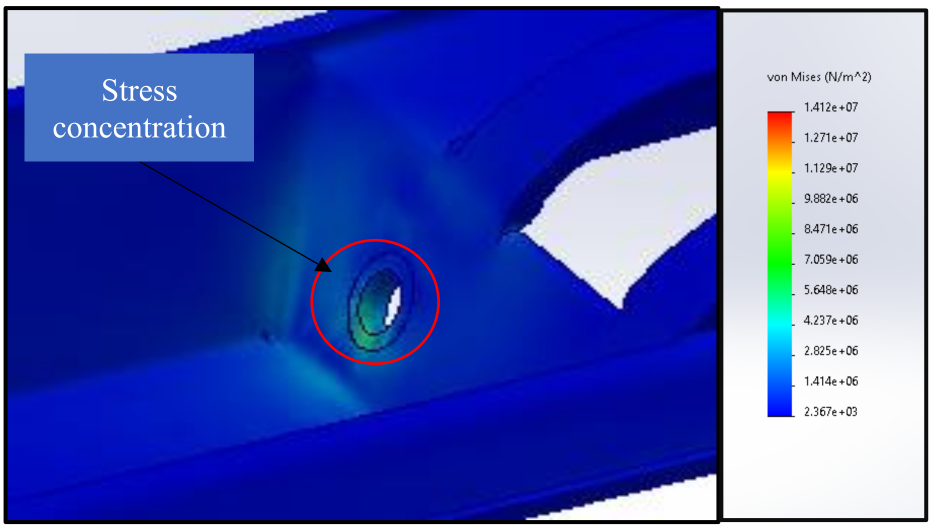


Fig.II.13: stress (Von Mises)

- Displacement Analysis: The minimum displacement recorded was 0.01 mm, indicated by the light blue colour in the figure. This suggests areas within the part where no displacement occurred, indicating relative stability and minimal deformation under the applied forces. On the other hand, the maximum displacement observed was 1.295 mm, represented by the red colour. These regions signify significant deformation or displacement within the part, indicating areas of concern that may affect the functionality and performance of the handle.
  It is important to note that the maximum displacement of 1.295 mm corresponds to the screw that holds the part in place. This particular screw experiences the highest level of displacement due to the applied forces and serves as a critical point of analysis. Understanding the displacement of this screw is crucial for evaluating its stability and ensuring that it remains securely fixed within the assembly.
  The colour degradation in the figure followed a gradient from dark to light blue, then green, and finally yellow, with the maximum displacement displayed in red. This colour representation helps visualize the extent of deformation and

displacement across different regions of the part, with the red indicating the highest displacement.
Please refer to Figure II.14 (Displacement - Res disp) for a visual representation of the displacement distribution within the plastic part. This figure will provide further insight into the extent and locations of deformation and displacement within the part, with a specific focus on the maximum displacement of the screw that holds the part in place.

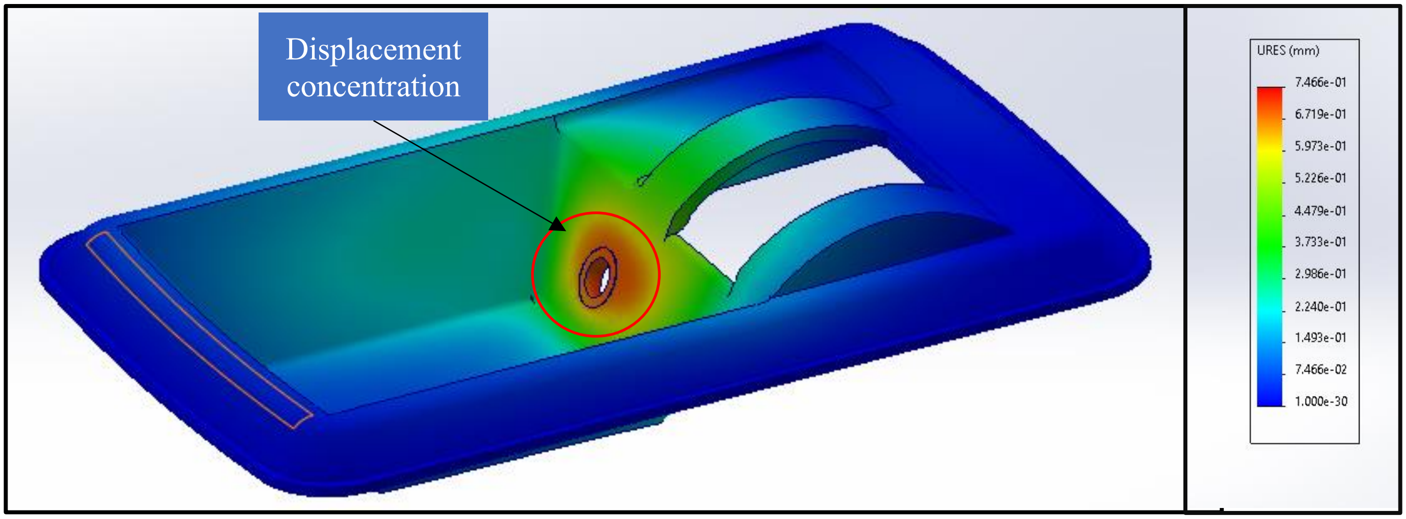


Fig.II.14: displacement (Res disp)

In summary, the stress and displacement analysis in this simulation provide essential insights into the structural behaviour of the handle when fixed onto the door. The information obtained allows us to identify areas of stress concentration and deformation, guiding us in optimizing the design, enhancing the part's reliability, and ensuring its long-term performance in real-world applications.

- **Thickness Analysis:** Thickness analysis involves evaluating the thickness distribution throughout the handle to ensure uniformity and structural integrity. It helps identify potential areas of weakness or excessive material, which can impact the handle's performance and durability. FEA techniques are often employed to simulate stress and deformation patterns and optimize the handle's thickness distribution.
  The thickness analysis of the plastic part was performed using the parameters in table 3:

Table 03: options used [7]

Parameter	**Value**
Analysis type	Thin analysis
Target thickness	2.5mm
Target thickness colour	/
Full colour range	On
Faces for local analysis	Full model
Resolution	Low

Thickness distribution in Fig.II.15 provides a visual representation of the thickness variation across the part, highlighting areas where the thickness falls within each range. The analysis reveals that the majority of the part has thickness values ranging from 1.27mm to 0.65mm, which covers 60.21% of the analysed area (more details in table 4). This suggests that the central portion of the part may have lower thickness compared to other regions.

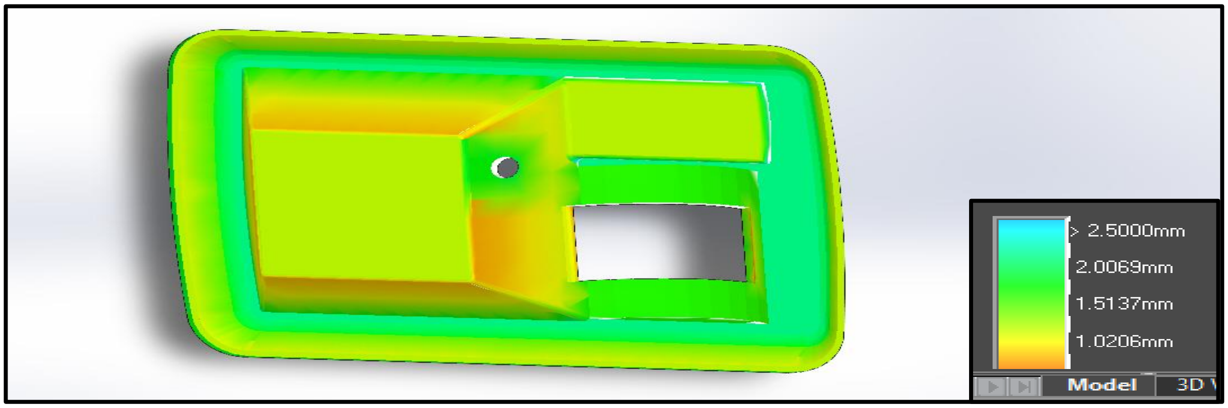


Fig.II.15: thickness distribution

Table 04: analysis details [9]

Thickness range	Number of faces	Surface area	% of analysed area
2.5mm to 1.88mm	3	2161.07 mm^2	14.89%
1.88mm to 1.27mm	25	2468.06 mm^2	17.00%
1.27mm to 0.65mm	35	8739.86 mm^2	60.21%
0.65mm to 0.03mm	30	601.27 mm^2	4.14%

The mass properties in table 5 indicate that the part has a surface area of 14531.51 mm^2, a volume of 8233.09 mm^3, and a mass of 184.0 grams. These properties provide insights into the overall size and weight of the part.

Table 05: mass properties [9]

Parameter	Value
Surface Area	14531.51 mm^2
Volume	8233.09 mm^3
Mass	184.0 grams

It is important to mention that the selected target thickness of 2.5mm for the thin analysis was based on previous test results. These previous tests, conducted with a thickness of 1mm, indicated that the part experienced inadequate structural integrity or potential failure. Therefore, the decision to increase the thickness to 2.5mm was made to address the observed issues and ensure the part's improved mechanical performance. The thickness analysis results provided valuable insights into the thickness distribution, highlighting areas where adjustments or reinforcement might be necessary for achieving the desired structural integrity and functionality of the plastic part.

- **Draft Analysis:**

Draft analysis assesses the draft angles of the handle's surfaces. Draft angles are crucial for ease of mould release during the injection moulding process. Insufficient draft angles can result in sticking or damage to the moulded part, while excessive draft angles may cause improper fit or functionality. Advanced CAD software, such as SW **[9]**, allows designers to analyse and optimize draft angles for efficient mould release.

In this analysis, as Figure II.16 illustrate, a draft angle of 1 degree was selected and applied to the handle's surfaces. The results of the analysis indicated that the draft angles were successful in achieving optimal mould release. The part exhibited 100% positive draft on the upper side (as shown in Figure II.17), indicating that all surfaces in this area had sufficient draft angles to facilitate smooth ejection from the mould. Similarly, the part demonstrated 100% negative draft on the bottom side (as shown in Figure II.18), ensuring effective release without any sticking issues.

The presence of yellow faces in the analysis represents areas that require additional draft. However, it is mentioned that all these faces were successfully eliminated, suggesting that the draft angles were appropriately adjusted or modified to eliminate the need for additional draft in those areas.

These results are highly promising as they indicate that the selected draft angles and their distribution on the handle's surfaces are well-suited for efficient mould release. The achievement of 100% positive and negative draft indicates that the handle can be reliably manufactured through injection moulding without encountering any significant issues during the demoulding process

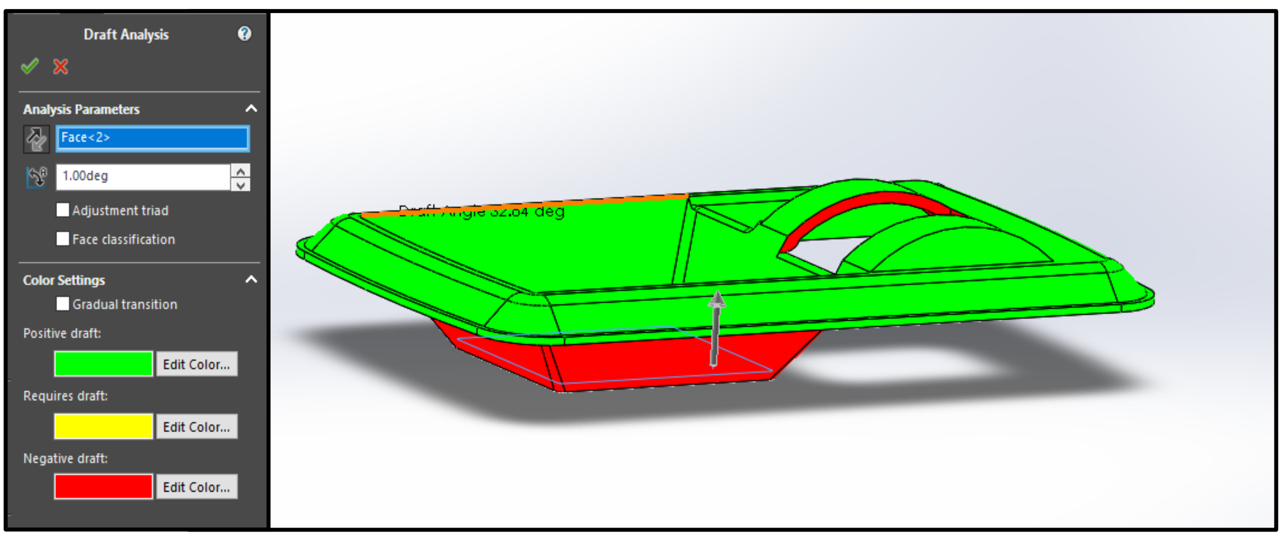


Fig. II.16: Draft analysis

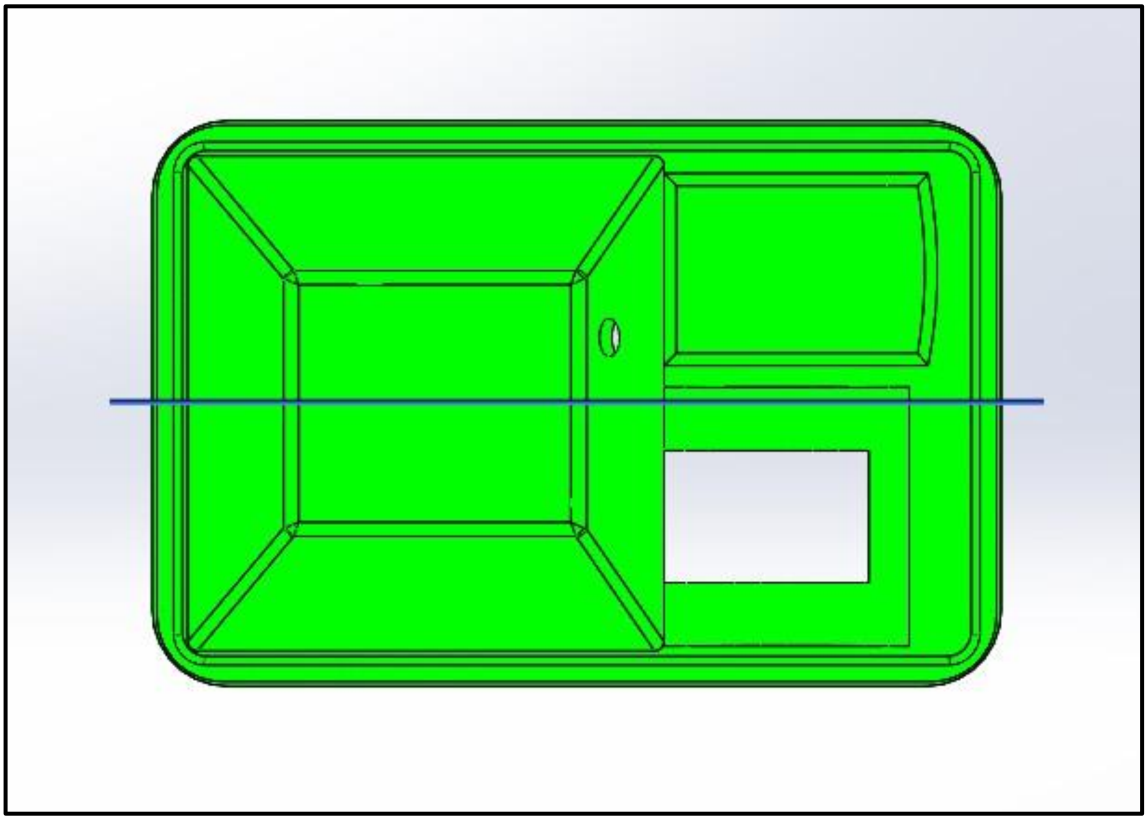


Fig. II.17: Draft analysis- top view

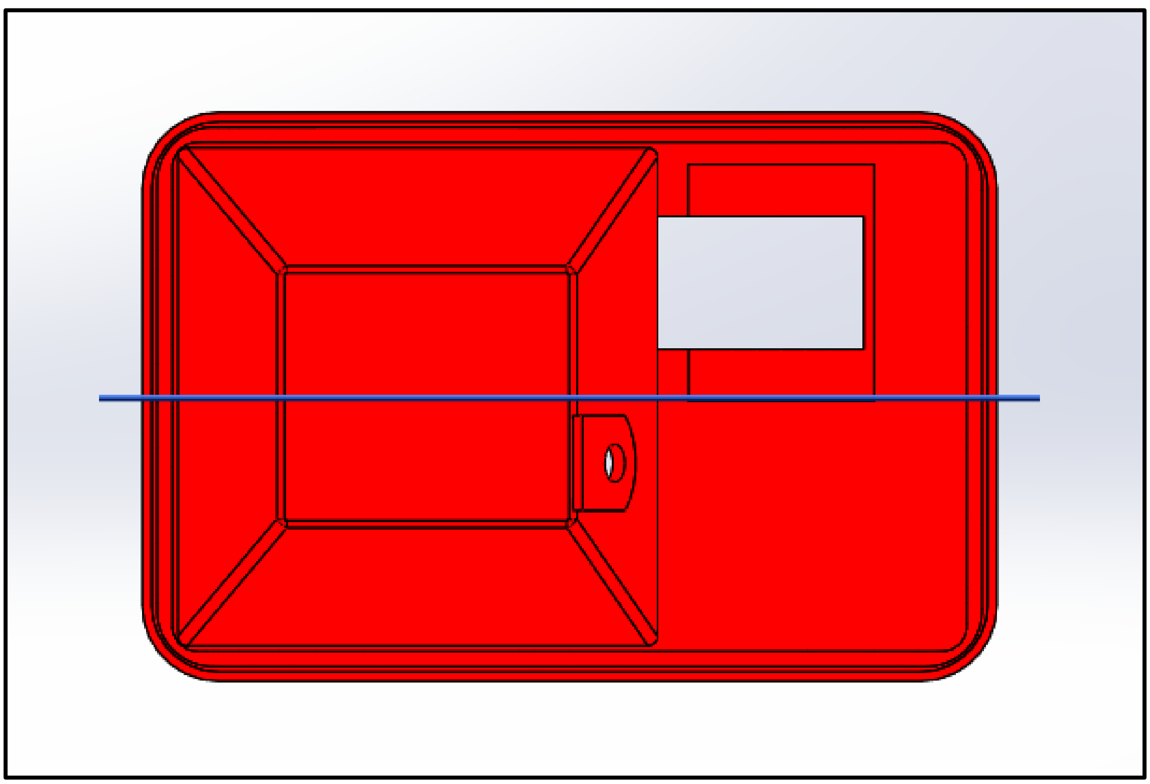


Fig. II.18: Draft analysis- bottom view

- **Parting Line and Moulding Direction:**

Parting line analysis involves determining the optimal location for the parting line—the separation line between the mould halves. The moulding direction is the path along which the molten plastic flows into the mould cavity. Analysing the parting line and moulding direction helps minimize cosmetic defects, optimize material flow, and enhance the overall quality of the moulded handle.

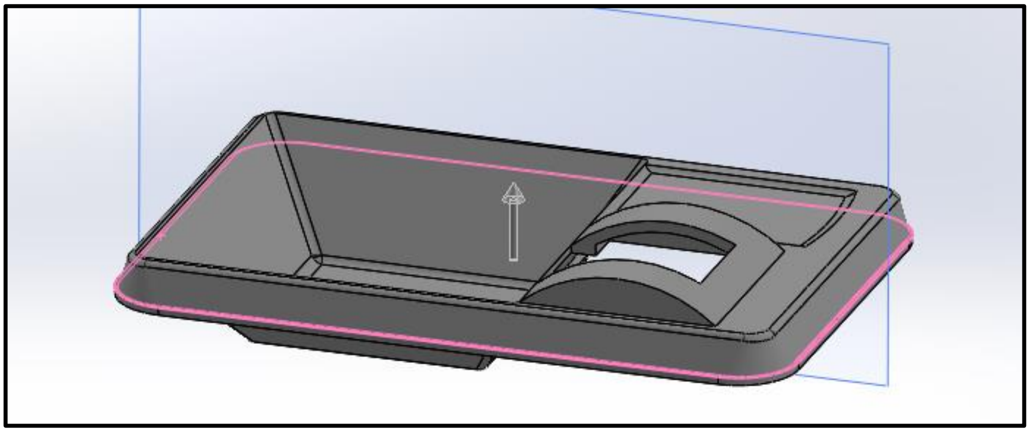


Fig.II.19: parting line and moulding direction

2.7 Conclusion

In conclusion, this chapter focused on several important aspects related to the analysis of a plastic handle using SW **[9]**.

The stress analysis provided valuable insights into the von Mises stress distribution, highlighting regions of high stress concentration that could potentially lead to failure or deformation. By understanding these stress levels, designers can optimize the handle's design to enhance its structural integrity and ensure its long-term performance.

The displacement analysis revealed the extent of deformation and displacement within the handle. Understanding the displacement patterns helps evaluate the handle's mechanical response and its interaction with the surrounding components. This information is crucial for identifying potential interference or misalignment issues and making necessary adjustments to ensure proper fit and functionality.

The thickness analysis shed light on the thickness distribution across the handle, indicating areas where the thickness falls within different ranges. This analysis provides insights into potential concerns related to structural integrity and highlights regions that may require reinforcement or adjustments to ensure optimal performance.

The analysis of draft angles assessed their suitability for efficient mould release during the injection moulding process. The successful achievement of 100% positive and negative draft angles indicates that the handle's surfaces are appropriately designed to facilitate smooth ejection from the mould, minimizing the risk of sticking or damage.

Overall, this chapter's analysis serves as a crucial foundation for refining the handle's design, enhancing its performance, and ensuring its successful production and functionality in real-world usage scenarios.

CHAPTER III:
Injection Simulation

3.1 Introduction:

Injection simulation plays a crucial role in the optimization and validation of the injection moulding process. By simulating the various aspects of the process, designers and engineers can analyse and optimize key parameters to ensure high-quality and efficient production. This chapter focuses on different aspects of injection simulation, including filling analysis, time, cooling water temperature…etc.

3.2 Simulation Analysis:

In the simulation analysis, there are several key components that contribute to a comprehensive evaluation of the part's performance. Each part plays a crucial role in understanding various aspects of the injection moulding process and the behaviour of the plastic component. Let's explore these parts in detail:

- **Material Selection:**

Selecting the appropriate material for simulation is crucial as it affects the part's mechanical properties, flow behaviour, and response to external loads. The material properties, such as viscosity, thermal conductivity, and mechanical strength, need to be defined accurately to obtain reliable simulation results. It is important to choose a material that closely represents the actual material being used in production.

Type:	Shell
Material	
Polymer:	ABS
Product:	(P) Generic material / Generic ABS
Melt Temperature:	230.00 (°C)
Mold Temperature:	80.00 (°C)
Ejection Temperature:	90.00 (°C)
State	
Volume:	8.23 (cm3)
Mass:	9.07 (G)
Size:	
X: 97.02 (mm)	
Y: 20.40 (mm)	
Z: 60.00 (mm)	

Fig.III.1: generic material (ABS) properties [9]

- **Injection Gate:**

The injection gate is a critical element that controls the flow of molten plastic into the mould cavity. Setting the injection gate involves selecting the appropriate location and size to ensure proper filling of the part while minimizing potential defects like air traps or flow marks. The gate's position and dimensions significantly impact the flow behaviour and overall quality of the moulded part.

Figure III.2, illustrating the gate position, provides a visual representation of how the molten plastic enters the mould cavity. The gate's location on the bottom face indicates that the plastic material is injected from underneath the part. This placement ensures that the molten plastic can reach all areas of the mould cavity effectively, minimizing the chances of voids or incomplete filling.

The chosen gate position reflects a careful consideration of the part's geometry and the desired flow behaviour during the injection moulding process. By selecting the gate in the middle of the part on the bottom face, it demonstrates a strategic decision to optimize the fill pattern and enhance the overall quality of the moulded part.

Additionally, it should be noted that the chosen gate position on the bottom face of the part, specifically in the middle, was also influenced by aesthetic considerations. By selecting this position, the intention was to minimize the appearance of the gate footprint on the visible surfaces of the part. Placing the gate on the bottom face helps to conceal any potential gate marks or blemishes that could affect the overall aesthetics of the final moulded part.

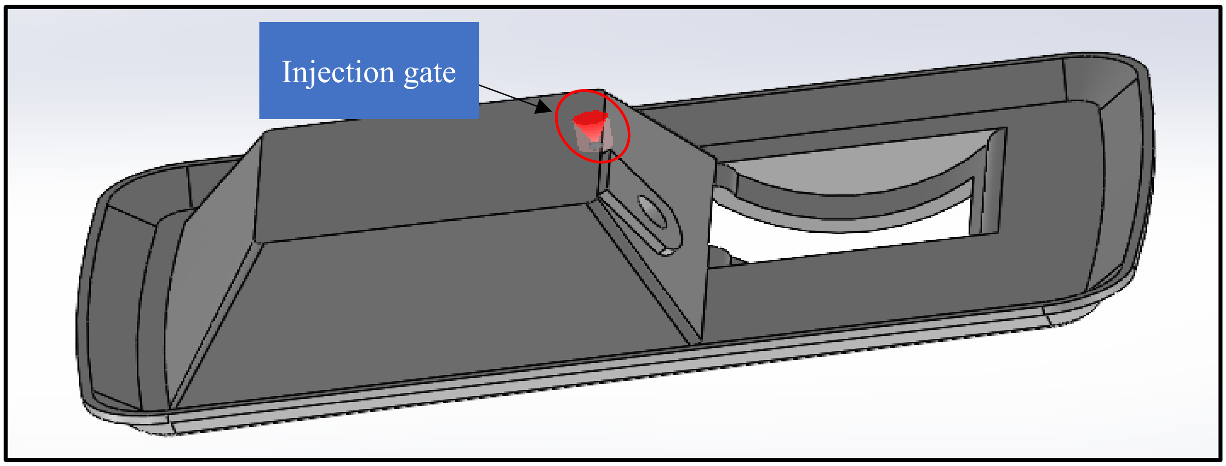


Fig.III.2: injection gate

- **Meshing**:

Meshing involves generating a mesh of small cells or elements that capture the geometry's details accurately. The mesh quality directly influences the accuracy and efficiency of the simulation results.

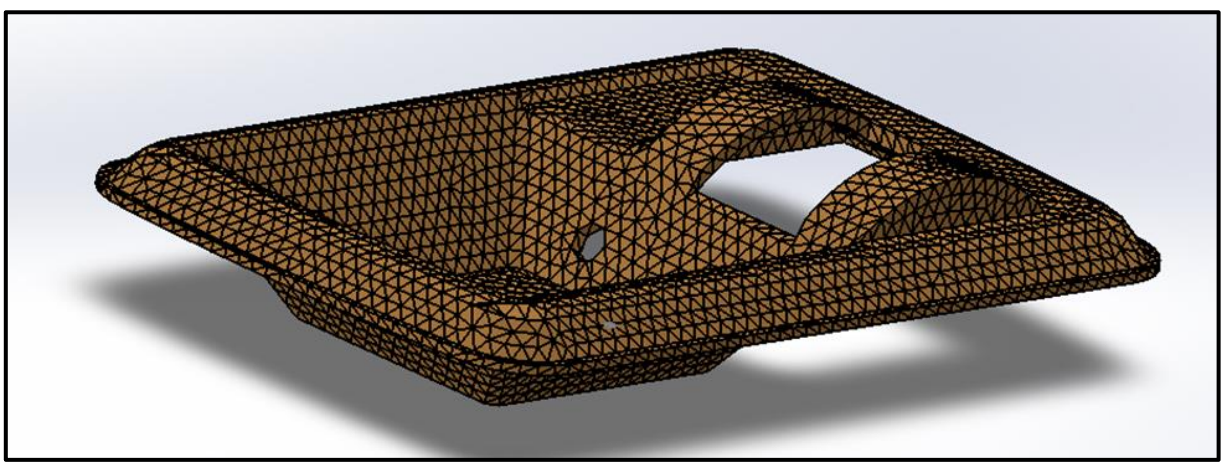


Fig.III.3: meshing

- **Boundary Conditions:**

Boundary conditions define the external loads and constraints applied to the part during the simulation. These include fixing certain areas of the model to simulate constraints and applying loads or pressures to simulate real-world operating conditions. Properly setting up the boundary conditions ensures that the simulation accurately represents the anticipated loading and operating conditions of the part.

- **Analysis Settings:**

This includes specifying parameters such as time steps, solver options, convergence criteria, and other simulation parameters. These settings control the accuracy, stability, and computational efficiency of the simulation. It is important to choose appropriate settings to achieve reliable results within a reasonable computational time.

- **Fill Analysis**

Filling analysis simulates the flow of molten plastic into the mould cavity during the injection moulding process. It provides insights into the filling pattern, flow front advancement, and potential flow-related issues such as flow imbalances, flow hesitation, or short shots. By analysing the filling behaviour, adjustments can be made to optimize gate locations, runner systems, and process parameters to achieve uniform filling and minimize defects.

Figure III.4 represents the filling process when 10% of the plastic has been injected into the mould cavity.

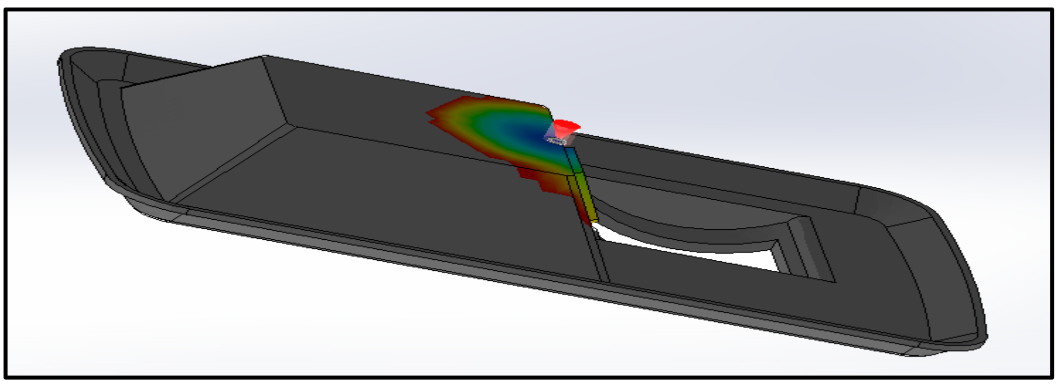


Fig.III.4: 10% of plastic injected

Figure III.5 shows the filling progress when 40% of the plastic has been injected. It allows for an evaluation of how the flow front has advanced and any potential flow-related issues that may have emerged during this stage.

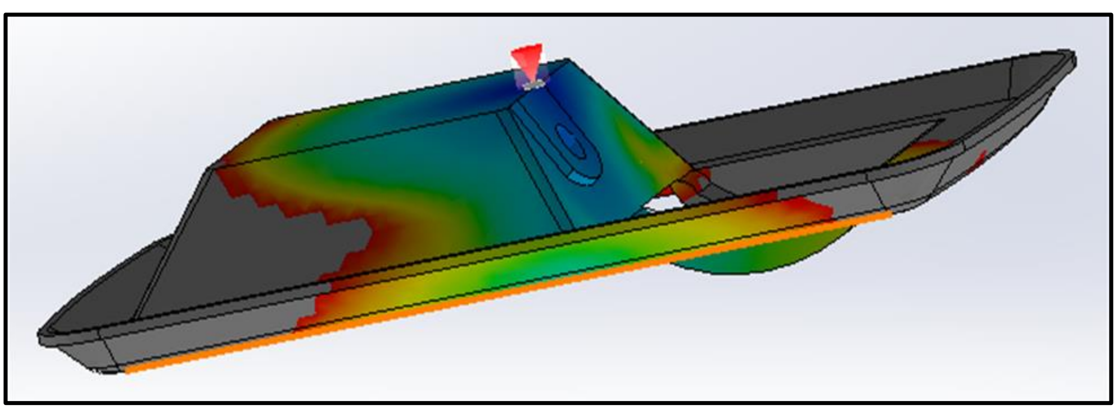


Fig.III.5: 40% of plastic injected

Figure III.6 provides a full path view of the filling process, illustrating the complete flow of the molten plastic from the gate to the furthest points within the mould cavity. This comprehensive view helps identify any potential flow imbalances or areas where the filling may have encountered challenges.

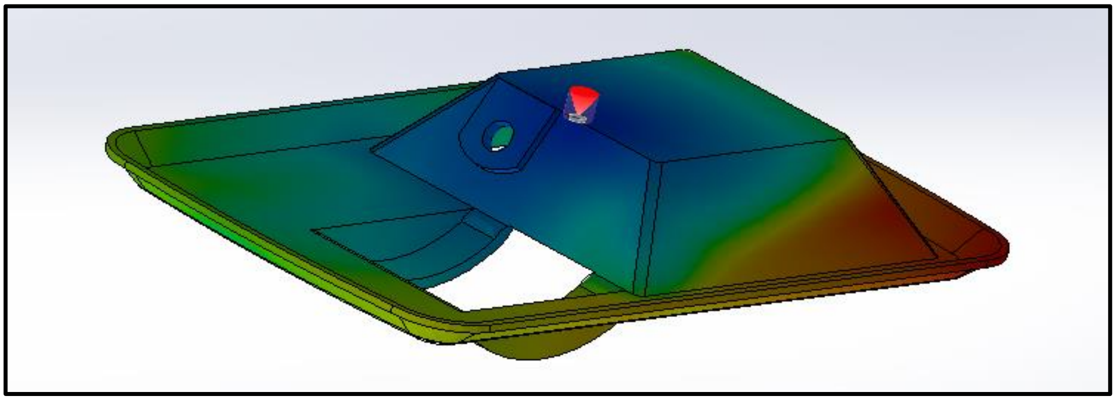


Fig.III.6: full path view

Figure III.7 presents the maximum inlet pressure observed during the filling analysis. The pressure gradually increased to reach a maximum of 70 MPa over a total injection time of 0.65 seconds. This information is crucial for assessing the pressure requirements of the injection moulding process and ensuring that the selected machine and equipment can handle the necessary pressures.

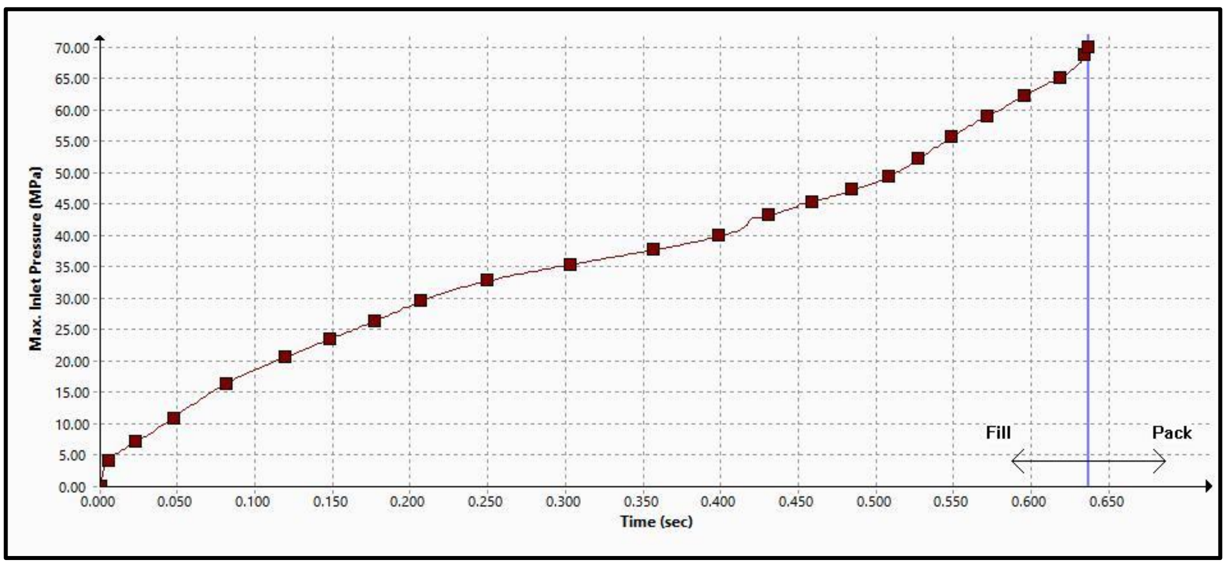


Fig.III.7: Max. inlet pressure

Figure III.8 depicts the melt front flow rate throughout the injection process. It shows that the flow rate started at 12 cc/s, reached a maximum of 13 cc/s, and remained within this range during the entire injection time of 0.65 seconds. This information helps evaluate the flow behaviour and ensures that the molten plastic is being delivered consistently and at an appropriate rate.

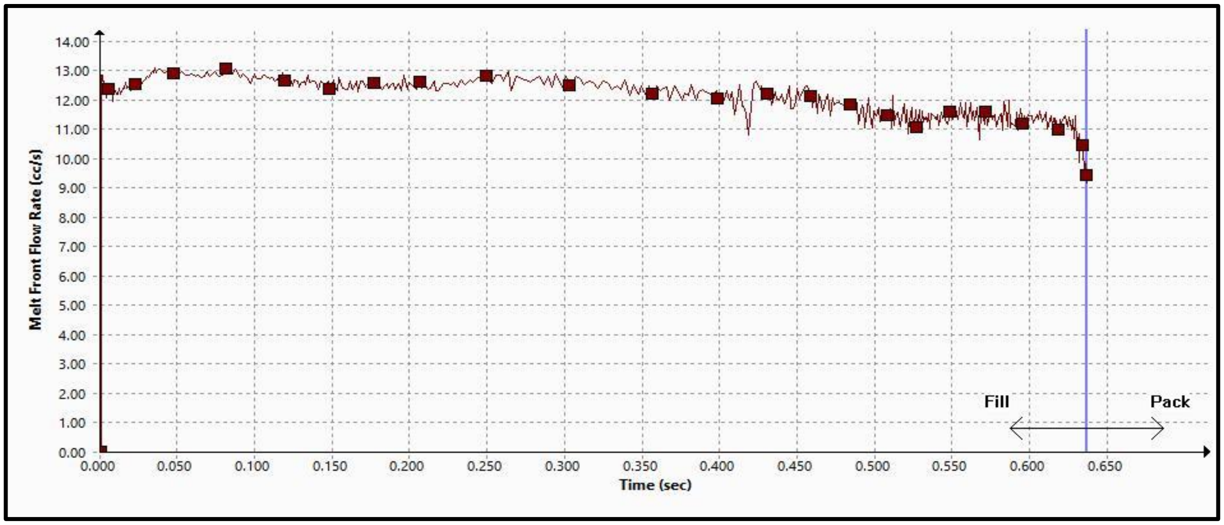


Fig.III.8: melt front flow rate

Figure III.9 provides insights into the part mass increase during the filling process. It indicates a gradual increase in the part's mass without facing any significant issues. However, a slight degradation in the filling is noticeable between 0.25 seconds and 0.4 seconds due to the small thickness in the borders of the part. This observation highlights a potential area for improvement, suggesting the need for adjustments in the mould design or process parameters to ensure more uniform filling.

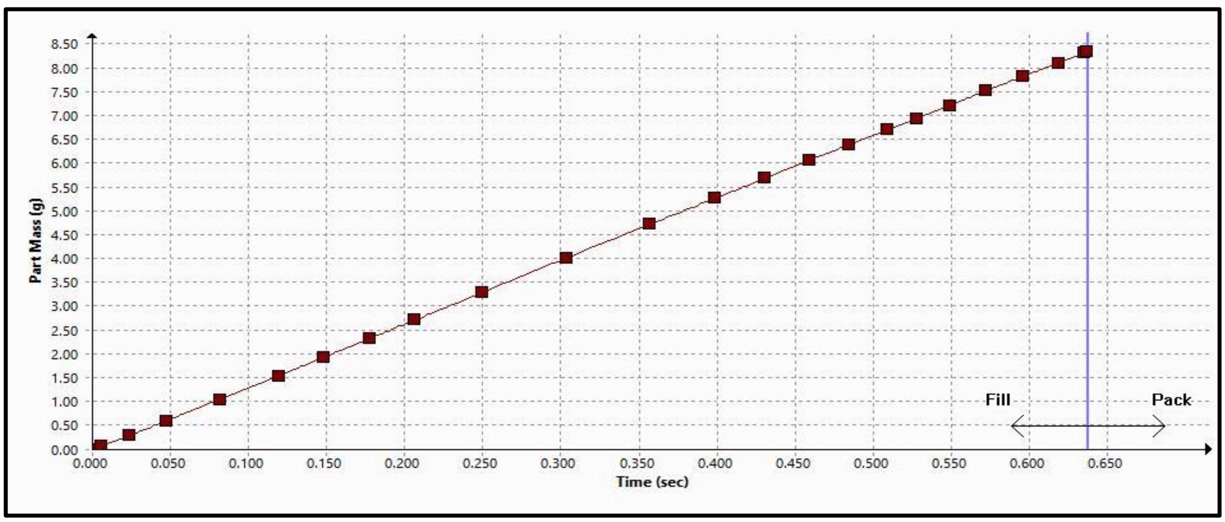


Fig.III.9: part mass

- **Fill Time**

The fill time plot displays the profile of the plastic melt as it flows through the mould part cavity during the filling stage of the injection moulding process. The blue regions indicate the start or beginning of the flow front. The red regions indicate either of the following:

1- The flow front position at any given time interval during an animation of the filling stage.

2- The end of fill when the flow has stopped, even if the software detects a short shot.

- **Pressure at End of the fill**

At the end of fill, the part has been successfully filled with an injection pressure of 54.2 MPa, the plastic in contact with the cavity wall freezes into a very thin frozen layer that has cooled down w mould. The thickness of this frozen layer is independent of the thickness of the part wall. The thickness is dependent on the melt and mould temperature differential and the material thermal conductivity.

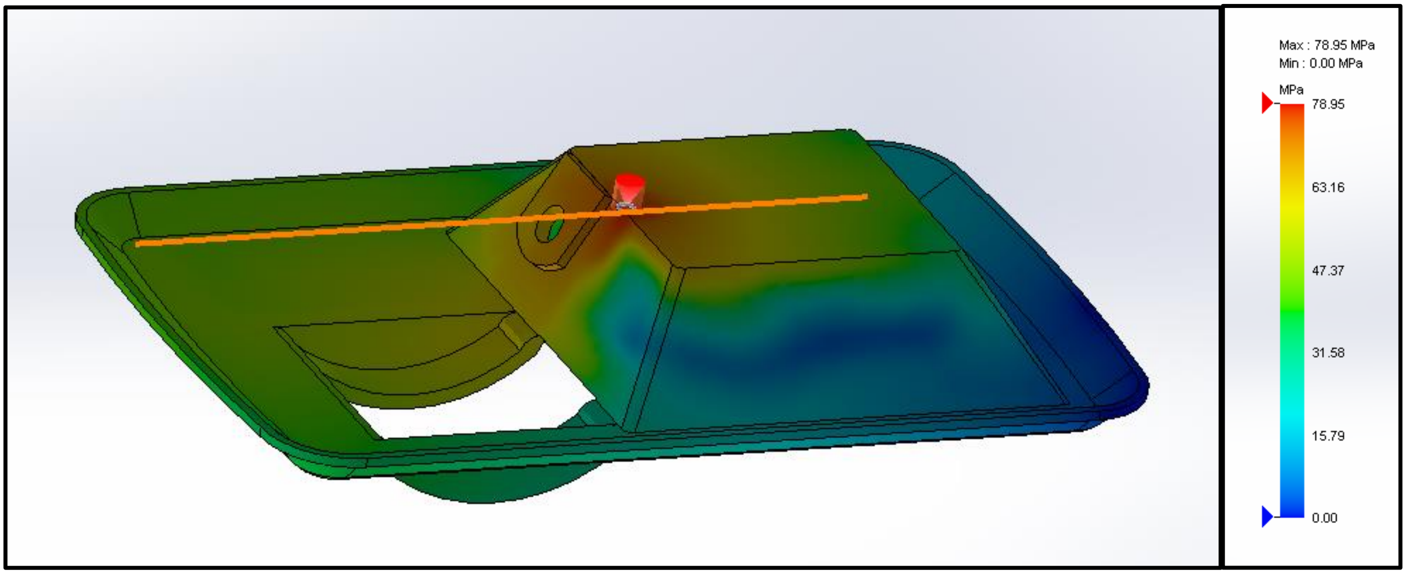


Fig.III.10: pressure view

On the outer surface of the part, the Temperature at End of Fill result displays temperature values taken from the centre of the solid mesh cell nearest to the surface. Since this location is slightly inside the part, temperatures will be higher than the mould wall temperature.

Note: This difference will decrease as creating a finer solid mesh with more cells through the thickness of the part.

It should be noted also that the time of the filling process mentioned in the analysis, which was 0.65 seconds, may appear relatively short. This is because the analysis focused on studying a mesh representation of the part using SW **[9]**. The time scale in the analysis may differ from the actual injection moulding process, which typically takes longer to complete.

While the time scale may differ, the analysis still provides valuable information about the flow behaviour and identifies potential areas for improvement. The focus is on understanding the flow characteristics, identifying any potential flow-related defects, and optimizing the design and process parameters accordingly.

- **Volumetric Shrinkage at the end of the fill**

The volumetric shrinkage at end of fill can indicate areas of potential concern. High rates of shrinkage will occur in thick sections of a plastic part which do not undergo a sufficient packing stage during the moulding process.

In Figure III.12, the volumetric shrinkage view illustrates the regions where significant shrinkage is anticipated. The areas indicated in yellow and red represent locations with higher levels of volumetric shrinkage. These regions require attention and adjustment in the molding process to ensure proper packing and minimize excessive shrinkage.

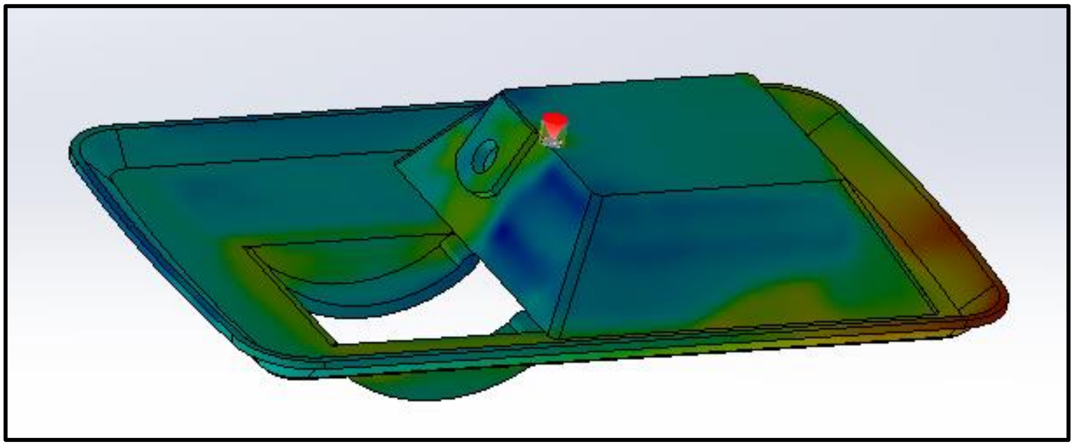


Fig.III.11: volumetric shrinkage view

It is important to mention that the shrinkage value of 1.5 was chosen in the next chapter during the core and cavity design phase. This specific shrinkage value was selected based on the material properties and requirements of the part. By incorporating this chosen shrinkage value into the core and cavity design, it ensures that the final moulded part will compensate for the anticipated shrinkage during the cooling and solidification process, resulting in the desired dimensions and dimensional accuracy.

- **Weld lines**

Weld lines are formed when two or more plastic melt flow fronts come together and they can be caused by mould shut-off surfaces, mould core features, multiple injection locations or wall thickness variations that cause flow front promotion or hesitation. Weld lines generally form 180° opposite of the point where the melt front makes contact with the standing core of a shut-off surface.

Figure III.12, representing the weld lines view, provides a visual representation of the distribution of weld lines across the part. The presence and intensity of weld lines can vary based on factors such as part geometry, material properties, and injection moulding parameters.

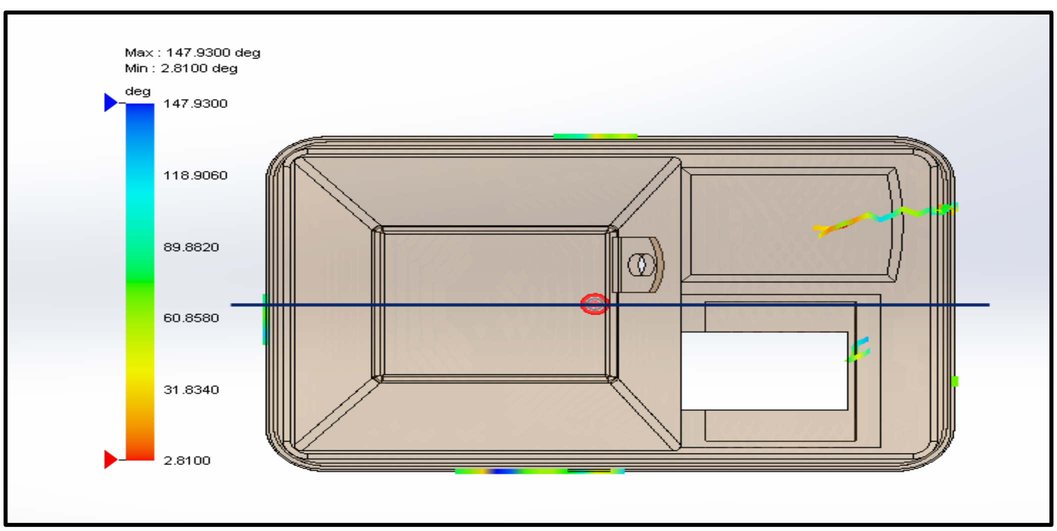


Fig.III.12: Weld lines view

The observation of a maximum value of 148 degrees indicates a localized area where the molten plastic encountered obstacles or features within the mould cavity, resulting in a significant fusion of flow fronts. This high temperature region may require attention, as it could potentially affect the part's structural integrity or surface appearance.

The presence of smaller areas with weld lines ranging between 60 and 90 degrees suggests the fusion of flow fronts in these regions as well, albeit to a lesser extent. While not as prominent as the maximum value area, these regions should still be evaluated and, if necessary, addressed to ensure optimal part quality.

It is worth noting that areas with temperatures below 31 degrees, where weld lines are barely visible, indicate a more favourable flow behaviour and improved weld line formation. This suggests that the molten plastic flowed smoothly and without significant obstructions or fusion in these regions.

Overall, understanding the distribution of weld lines through visual analysis aids in optimizing the injection moulding process, reducing weld line formation, and achieving high-quality moulded parts with enhanced structural integrity and improved surface aesthetics.

3.3 Conclusion:

The results obtained from the initial injection simulation analysis provide a promising foundation for the assessment of the part design and manufacturing process. The analysis reveals important insights and considerations that can guide further exploration and optimization.

The injection simulation demonstrates that the part can be successfully filled with an injection pressure of 54.2 MPa, which is well below the specified maximum limit, indicating a margin of safety in terms of pressure requirements. Additionally, the results indicate that the material remains within an acceptable temperature range, minimizing the risk of plastic material degradation during the filling process.

However, the analysis also highlights a concern regarding the minimum flow front temperature, which falls significantly below the starting melt temperature. This cooling effect can potentially lead to challenges such as filling and packing issues, increased injection pressure requirements, and compromised weld line integrity and appearance. It is crucial to address and mitigate these concerns in subsequent analyses to ensure optimal part quality and performance.

On a positive note, the flow front melt temperature remains within an acceptable range of variation from the starting melt temperature. This suggests that the mould filling and packing stages are likely to be successful, reducing injection pressure requirements and increasing the probability of producing a part with desirable properties.

The predicted cooling time, which identifies when 90% of the part temperature reaches below the material ejection temperature, offers valuable information for optimizing the cooling process during manufacturing. This data can be utilized to refine cooling strategies and enhance overall production efficiency.

CHAPTER IV:
Mould Design

CHAPTER IV*: Mould Design*

4.1 Introduction:

Mould design plays a crucial role in the manufacturing process of plastic parts. It involves the creation of custom-made moulds or dies that enable the precise replication of part geometries. An effective mould design ensures the production of high-quality parts with the desired specifications and characteristics.

In this chapter, we will delve into the intricate world of mould design, focusing on the specific requirements and considerations for designing the mould for the Suzuki Samurai interior door handle. The mould design process involves careful analysis, planning, and implementation of various factors to ensure the successful production of the desired part.

Throughout this chapter, we will explore the fundamental aspects of mould design, including parting line selection, core and cavity design, cooling system design, and the integration of additional mould components. We will also discuss the importance of considering manufacturing constraints, material properties, and production efficiency during the design process.

By understanding the principles and techniques of mould design, we can optimize the manufacturing process, reduce costs, and achieve the desired part quality. Through the application of industry best practices and the utilization of advanced software tools such as SW 2022, we can effectively design the mould for the Suzuki Samurai interior door handle, ensuring its functionality, durability, and aesthetic appeal.

Note: In this chapter on mould design, it is important to mention that we have intentionally chosen a relatively simple part, such as a car door handle cover, in order to simplify the work process and save time considering our limited experience in mould design. By focusing on a simpler part, we aim to effectively demonstrate the key principles and techniques involved in mould design while providing a solid foundation for future projects and expanding our expertise in this field.

4.2 Mould rules

In the process of designing the mould for injection moulding, several rules need to be followed to ensure optimal part quality and manufacturability. The following mould design rules are essential considerations to achieve successful injection moulding:

- **Parting Line and Mould Split:**

The parting line, which separates the mould into two halves, should be carefully determined. It should align with desired part features and minimize the need for additional cosmetic finishing. A well-designed mould split ensures accurate part replication and easy mould release.

- **Draft Angle:**

Incorporating draft angles on the vertical walls of the part is crucial for smooth ejection from the mould. The recommended draft angle depends on the material and surface texture. Insufficient draft angles can result in part sticking or damage during ejection.

- **Wall Thickness:**

Designing the mould cavity and core with consideration for the desired part's wall thickness is essential. Uniform wall thickness promotes proper material flow, reduces cycle time, and prevents defects such as sink marks or warping. Avoid sharp transitions or abrupt changes in wall thickness that could hinder material flow.

- **Fillets:**

Adding fillets at sharp corners of the mould helps distribute stress and prevent stress concentration. Sharp corners can lead to stress-induced failures, part deformation, or cosmetic defects. Properly designed fillets and radii enhance part strength and aesthetics.

- **Cooling System:**

Integration of an efficient cooling system within the mould is crucial for temperature regulation during the injection moulding process. Proper cooling ensures control over part quality, minimizes cycle time, and improves overall productivity. Utilize advanced cooling techniques or conformal cooling channels to optimize cooling efficiency.

- **Ejector System:**

Design a reliable ejector system to facilitate the smooth removal of the moulded part from the mould cavity. Proper positioning and sizing of ejector pins or plates ensure gentle ejection without causing any damage to the part.

- **Runner and Gate Design:**

Develop an efficient runner and gate system to ensure proper material flow into the mould cavity. Optimize the runner size, shape, and gate placement to minimize flow-related defects and ensure consistent filling. Consider the part geometry, material characteristics, and process parameters when designing the runner and gate system.

- **Venting:**

Incorporate proper venting in the mould design to allow the escape of trapped air or gases during the injection process. Sufficient venting prevents part defects such as burning, voids, or incomplete filling. Place vent locations strategically to avoid cosmetic surfaces.

- **Mould Material Selection:**

Select appropriate mould materials based on factors such as part volume, material properties, and expected production runs. Common mould materials include tool steels, pre-hardened steels, and aluminium alloys. Choosing the right material ensures mould durability, dimensional stability, and longevity.

4.3 Mould Functions

- **Clamping Function:**

SW enables the creation of robust clamping mechanisms to ensure proper alignment and secure closure of the mould halves during production.

- **Feeding Function:**

The software facilitates the design of efficient runner systems, gates, and sprues to control the flow of molten plastic into the mould cavity.

- **Regulation Function:**

SW allows for the precise design of flow channels and features that regulate the flow of plastic, ensuring uniform filling and minimizing defects.

- **Guiding and Positioning Function:**

With SW, it is possible to incorporate guiding and positioning elements, such as alignment pins and bushings, to ensure accurate assembly and alignment of the mould components. [10]

4.4 Core and Cavity Design

first step is to determine the parting line, which defines the boundary between the core and cavity sides of the mould. This step involved analysing the part geometry and identifying the optimal location for the parting line. Once the parting line was established, I selected the appropriate parting surface, ensuring a smooth and precise separation between the two halves of the mould.

Using CAD software, I created the core and cavity components based on the part geometry. This process involved translating the part design into a 3D model, incorporating features such as ribs, bosses, and other necessary details. I paid careful attention to draft angles, ensuring that the design allowed for easy part ejection and minimized the risk of mould damage.

Considering draft angles and undercuts, I evaluated the part design for any features that might hinder part ejection. For undercuts or complex features, I implemented suitable techniques such as side cores or collapsible cores to address these challenges effectively.

To provide a comprehensive understanding of the core and cavity design, I created detailed 2D drawings, showcasing the components from various angles. These drawings illustrate the dimensions specifications necessary for the fabrication and assembly of the mould

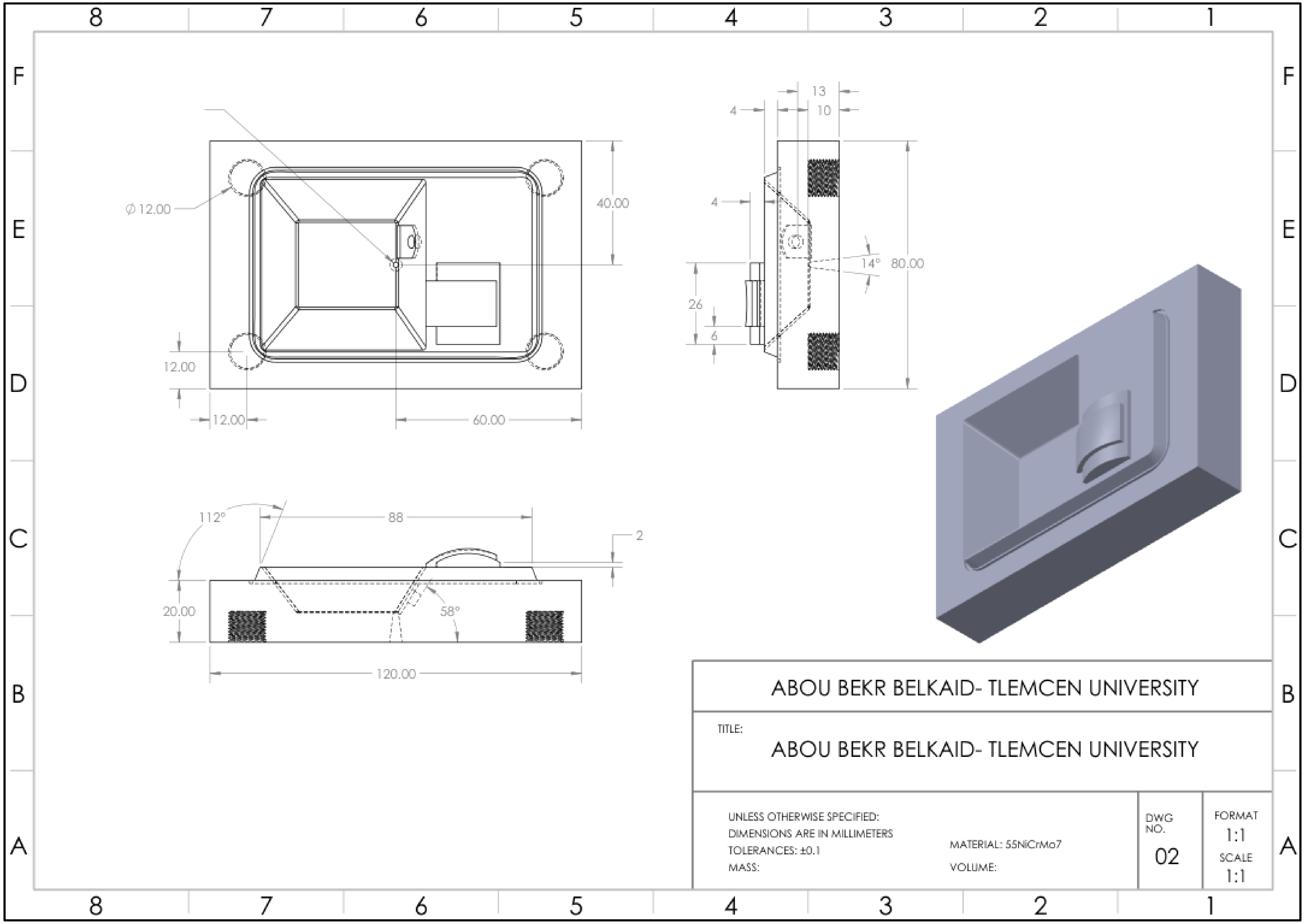


Fig. IV.1: SS door handle cavity- 2D drawing

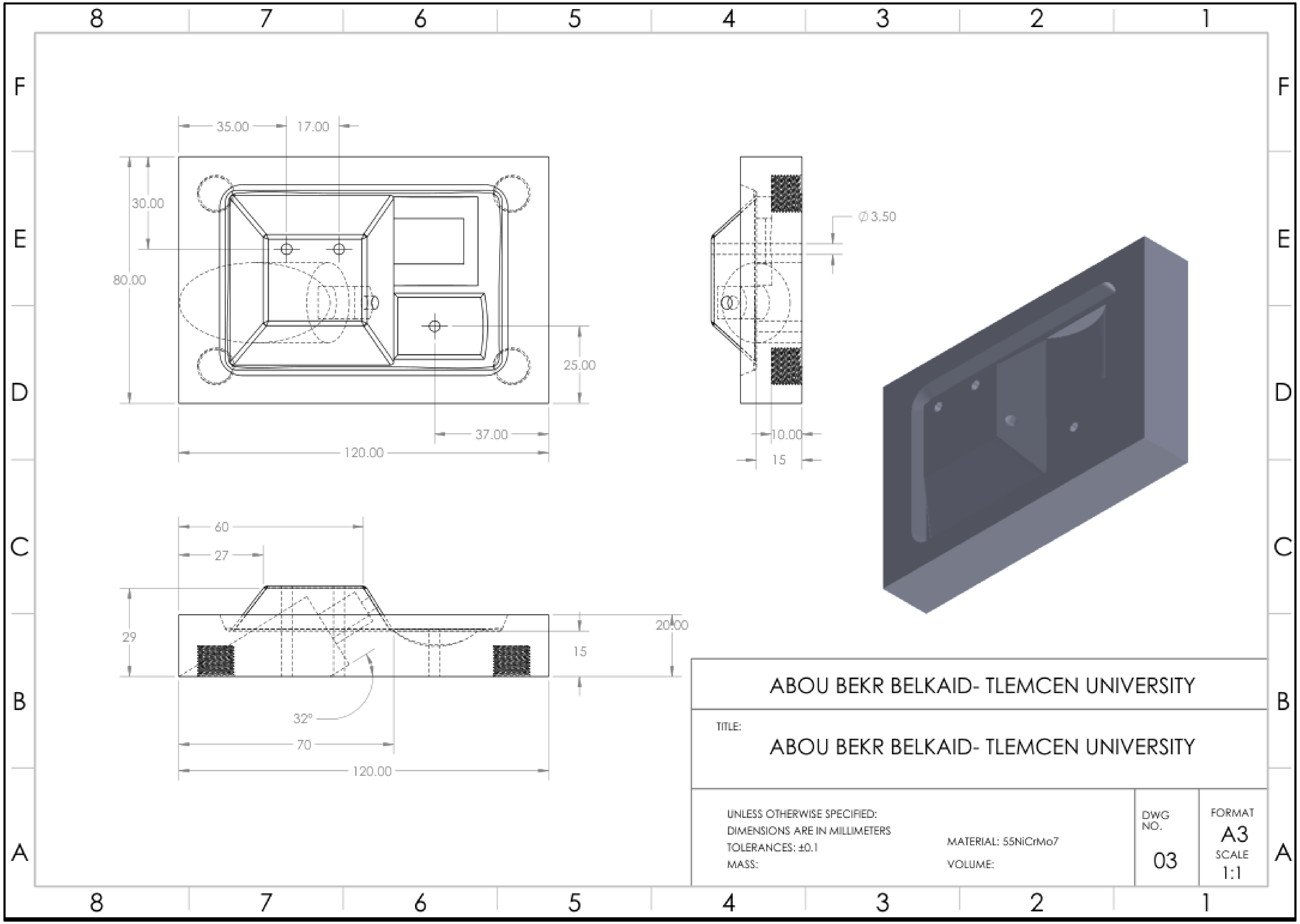


Fig. IV.2: SS door handle core- 2D drawing

Additionally, I generated 3D views that visually represent the assembled core and cavity within the mould, aiding in visualizing the final configuration.

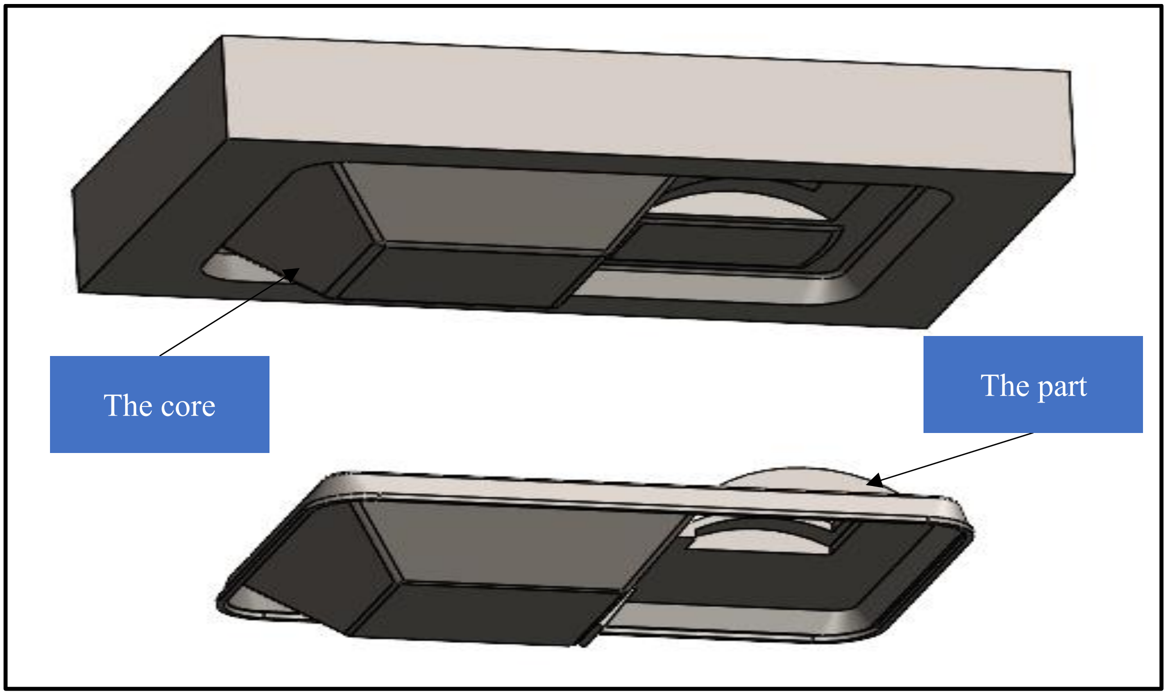


Fig. IV.3: SS door handle core- 3D view

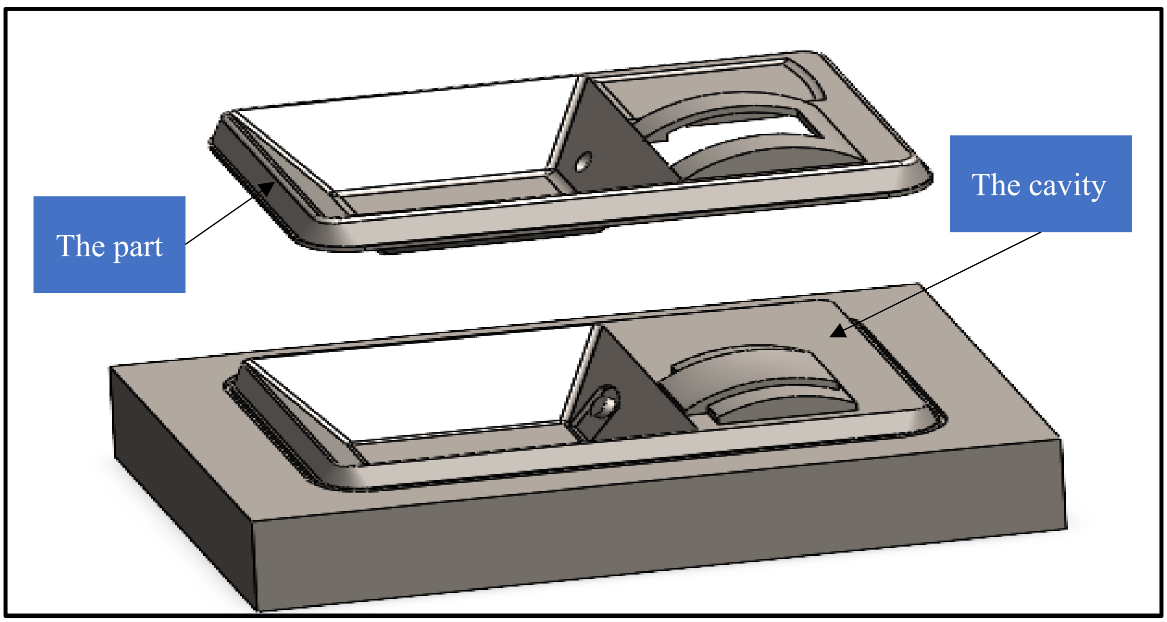


Fig. IV.4: SS door handle cavity- 3D view

The core and cavity design are critical for the moulding process, influencing the accuracy, efficiency, and quality of the final product. By meticulously considering the parting line, draft angles, cooling channels, and providing detailed documentation, I ensured the successful implementation of the mould design.

4.5 Fixed part of the mould

The fixed cavity block is designed to securely hold the moulded part within the mould cavity. It ensures proper clamping and stability during the injection moulding process.

- **Mould guide:**

Accurate positioning and alignment of mould components are essential for consistent part production and minimizing assembly errors. To assure this we use mould guides.

- **Support/fixation Plate:**

Support plates are crucial for withstanding the forces exerted during the injection and ejection processes. They provide structural integrity to the mould and ensure it can handle the high pressures and forces involved in moulding.

- **Cavity:**

The cavity is designed to accurately replicate the geometry and features of the interior car door handle cover.

- **Cooling System:**

Efficient cooling is essential for the injection moulding process. The cooling system design includes the creation of cooling channels within the mould.

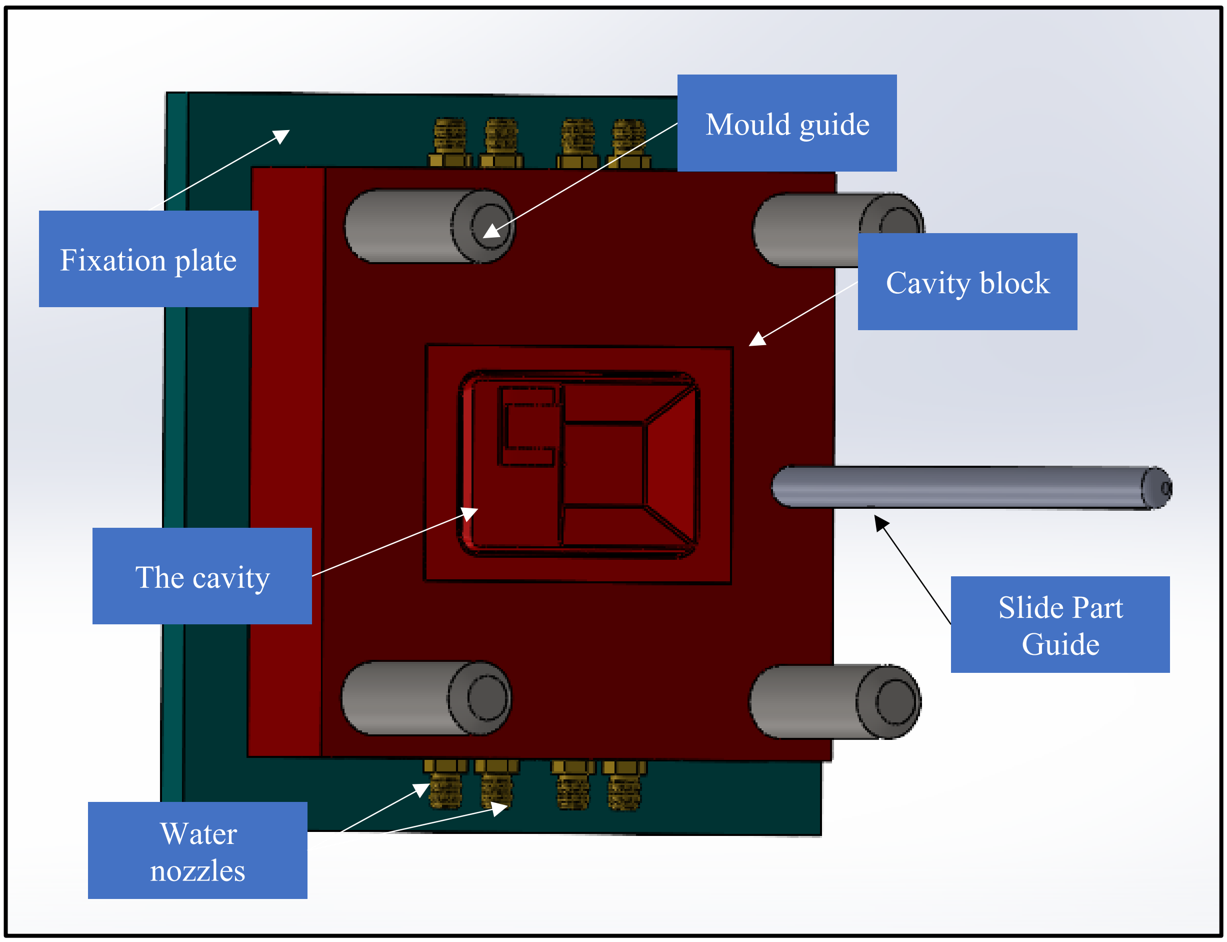


Fig. IV.5: mould assembly- fixed part

4.6 Movable part of the mould

The movable part is responsible for facilitating the ejection of the moulded part from the core and cavity, ensuring smooth and efficient operation during the moulding process. I have included various components in the movable part assembly to accomplish this task effectively.

- **Support/fixation Plate:**

The support plate provides a sturdy base for the core components and ensures their proper alignment within the mould. It helps maintain the stability and integrity of the core during the injection and ejection stages.

- **Ejectors Fixation Plate:**

The ejectors fixation plate securely holds the ejectors in place. It provides a rigid connection to ensure that the ejectors move synchronously and eject the part evenly from the core.

- **Ejectors Support:**

The ejectors support assists in guiding and supporting the ejectors during their movement. It helps maintain the alignment and stability of the ejectors, preventing any unwanted tilting or misalignment.

- **Ejectors:**

I have chosen to include four ejectors in the design to ensure efficient part ejection. The ejectors are positioned strategically to evenly distribute the ejection force and facilitate the smooth release of the part from the mould.

- **Ejector Block:**

The ejector block is located on each side of the mould and serves as a mounting point for the ejectors. It provides a secure attachment for the ejectors, allowing them to move in unison and effectively eject the part.

- **Ejector Guides:**

The ejector guides assist in guiding the ejectors during their movement. They ensure that the ejectors move in a straight and controlled manner, preventing any lateral deviation that could impact the ejection process.

- **Spring Return for Ejectors:**

The spring return mechanism is integrated with the ejector guides to provide the necessary force for the ejectors to retract back into their original positions after part ejection. This mechanism ensures that the ejectors are ready for the next cycle of the moulding process.

- **Ejector Front Closure:**

The ejector front closure is responsible for enclosing the front end of the ejector system. It helps maintain the alignment and positioning of the ejectors, ensuring their smooth operation during part ejection.

- **Core Block:**

The core block houses the core of the part and provides the necessary support and stability during the moulding process. It ensures accurate positioning of the core within the mould and facilitates its movement during part ejection.

- **Cooling System:**

The cooling system is an integral part of the movable section, as it helps regulate the temperature of the core components. It consists of cooling channels that allow the circulation of a cooling medium, effectively dissipating heat and promoting uniform cooling of the core.

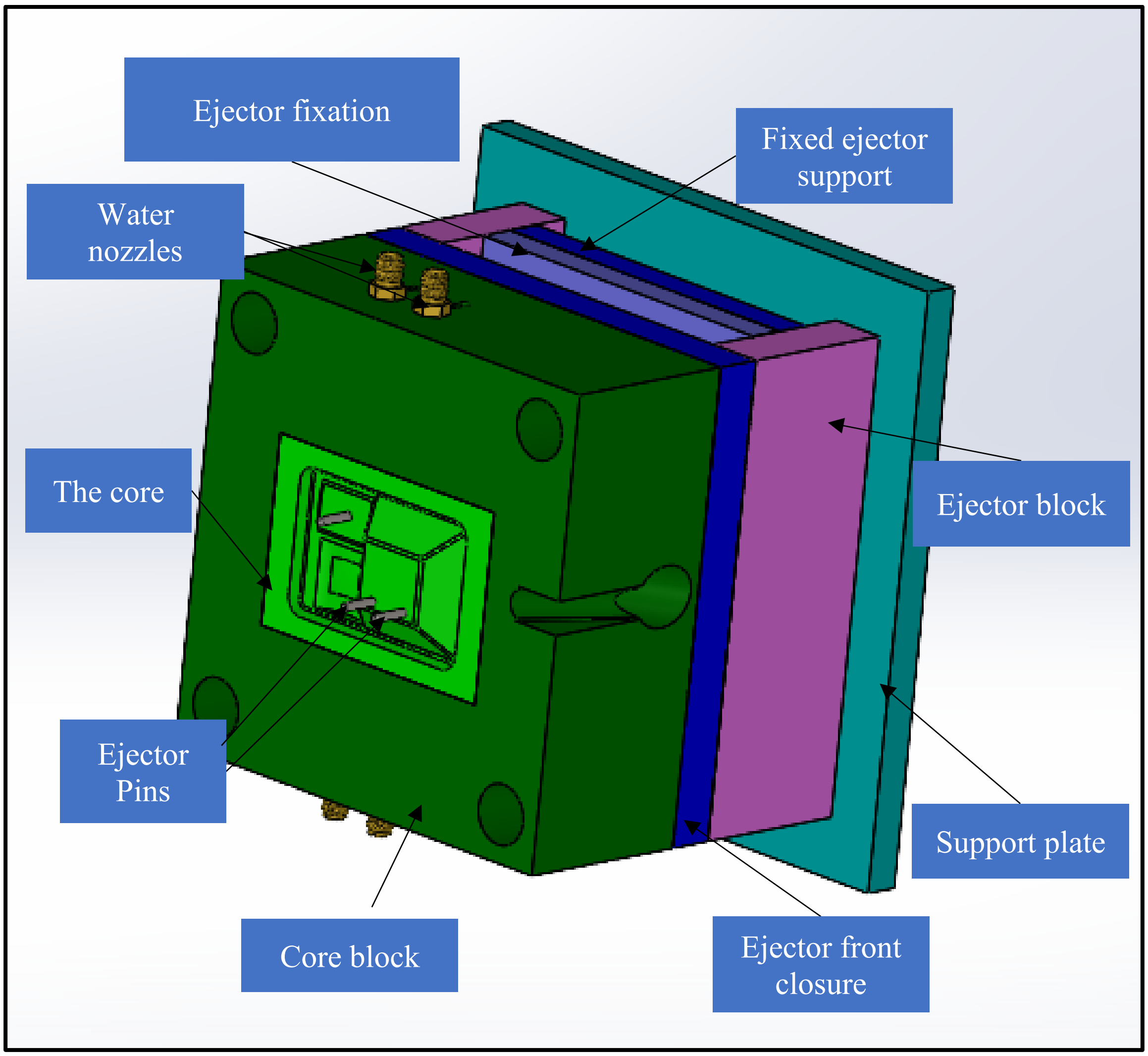


Fig. IV.6: mould assembly- movable part

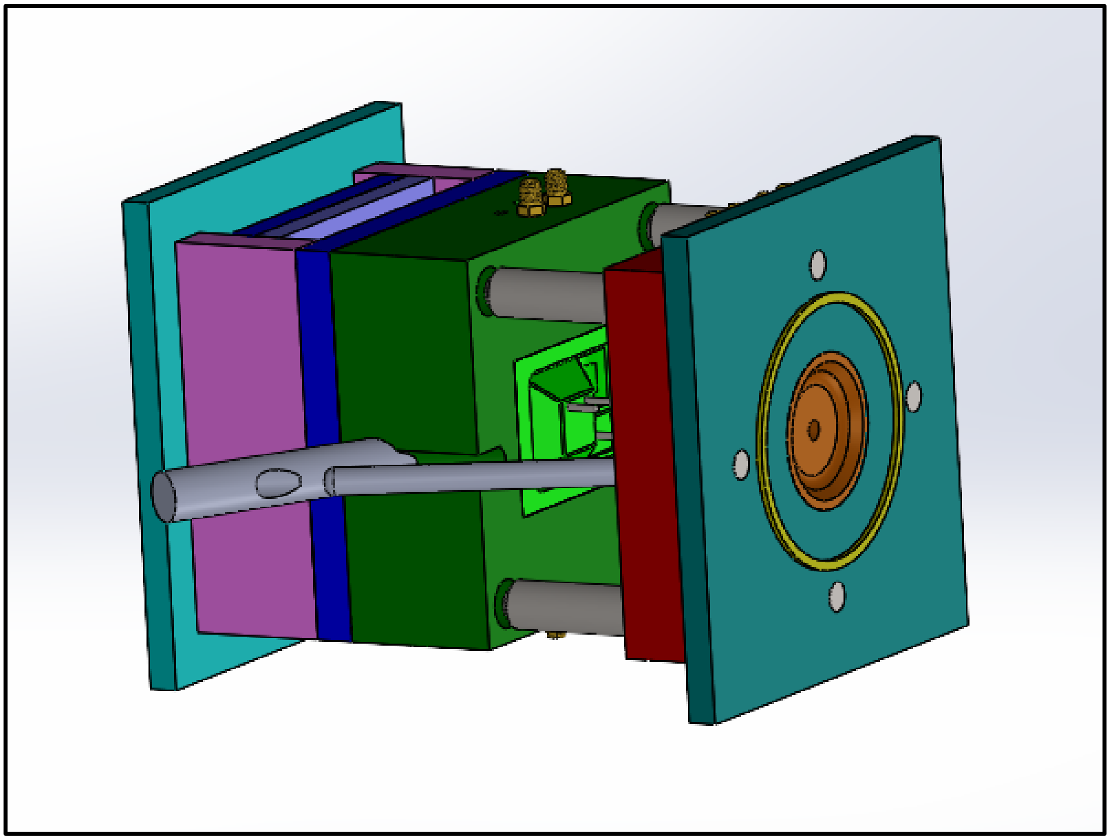


Fig. IV.7: mould assembly- view 1

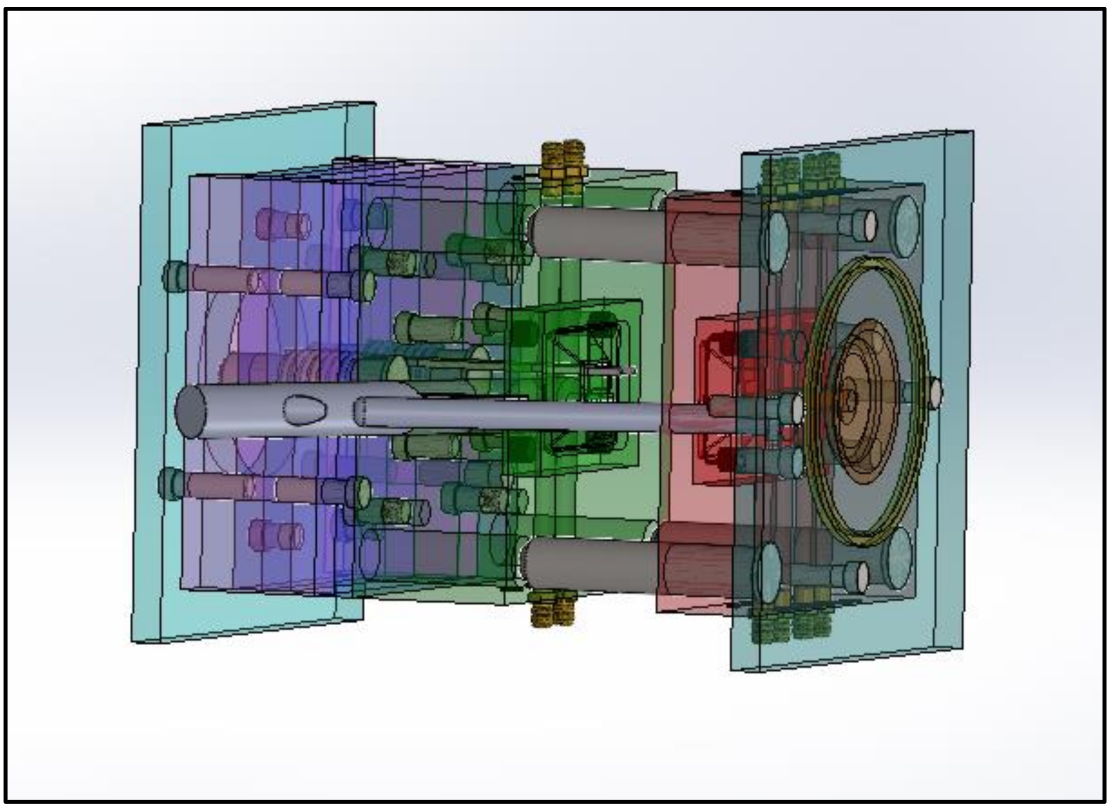


Fig. IV.8: mould assembly- view 2

4.7 Assembly Drawing + Detail Drawing

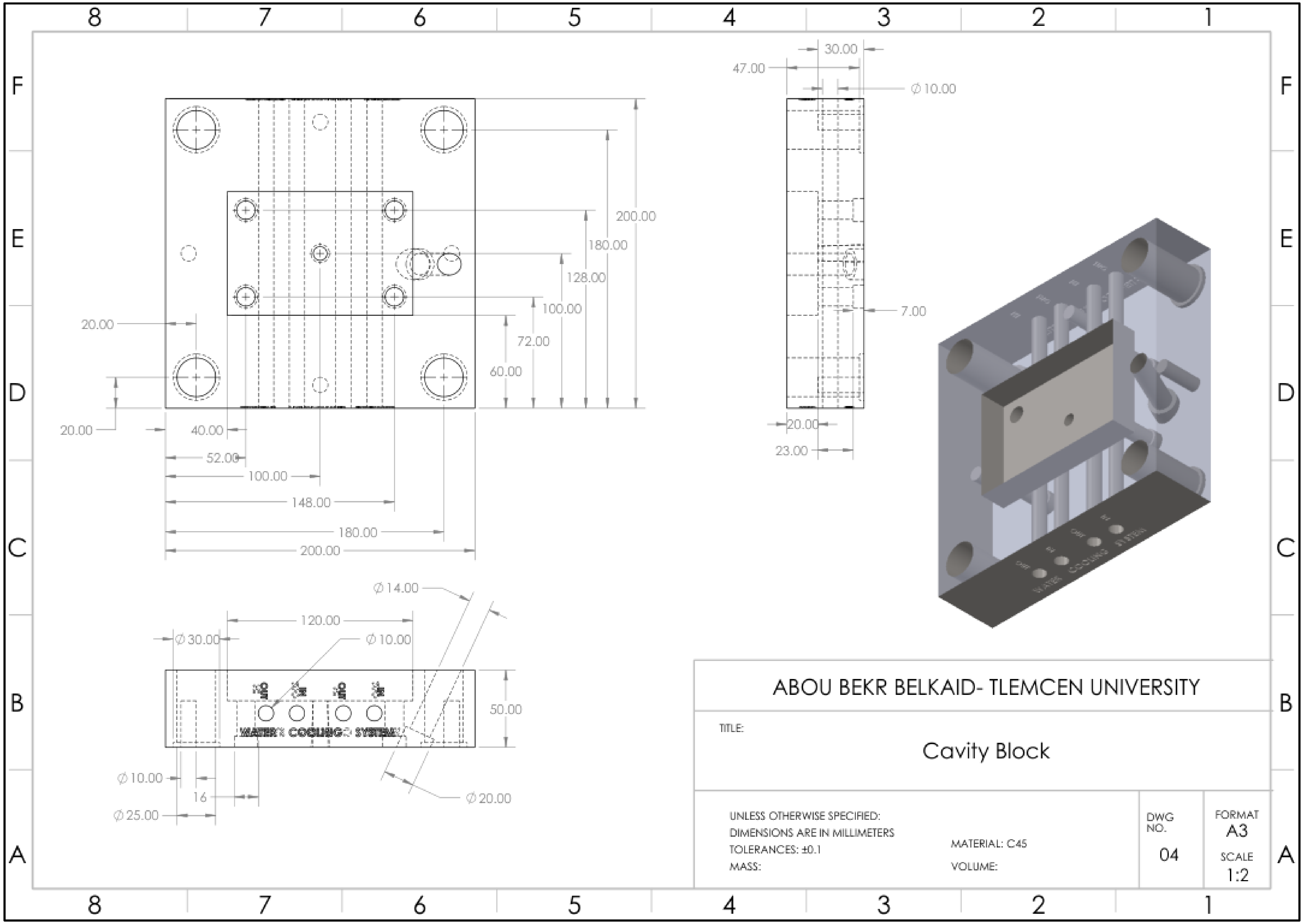


Fig.IV.9: cavity block

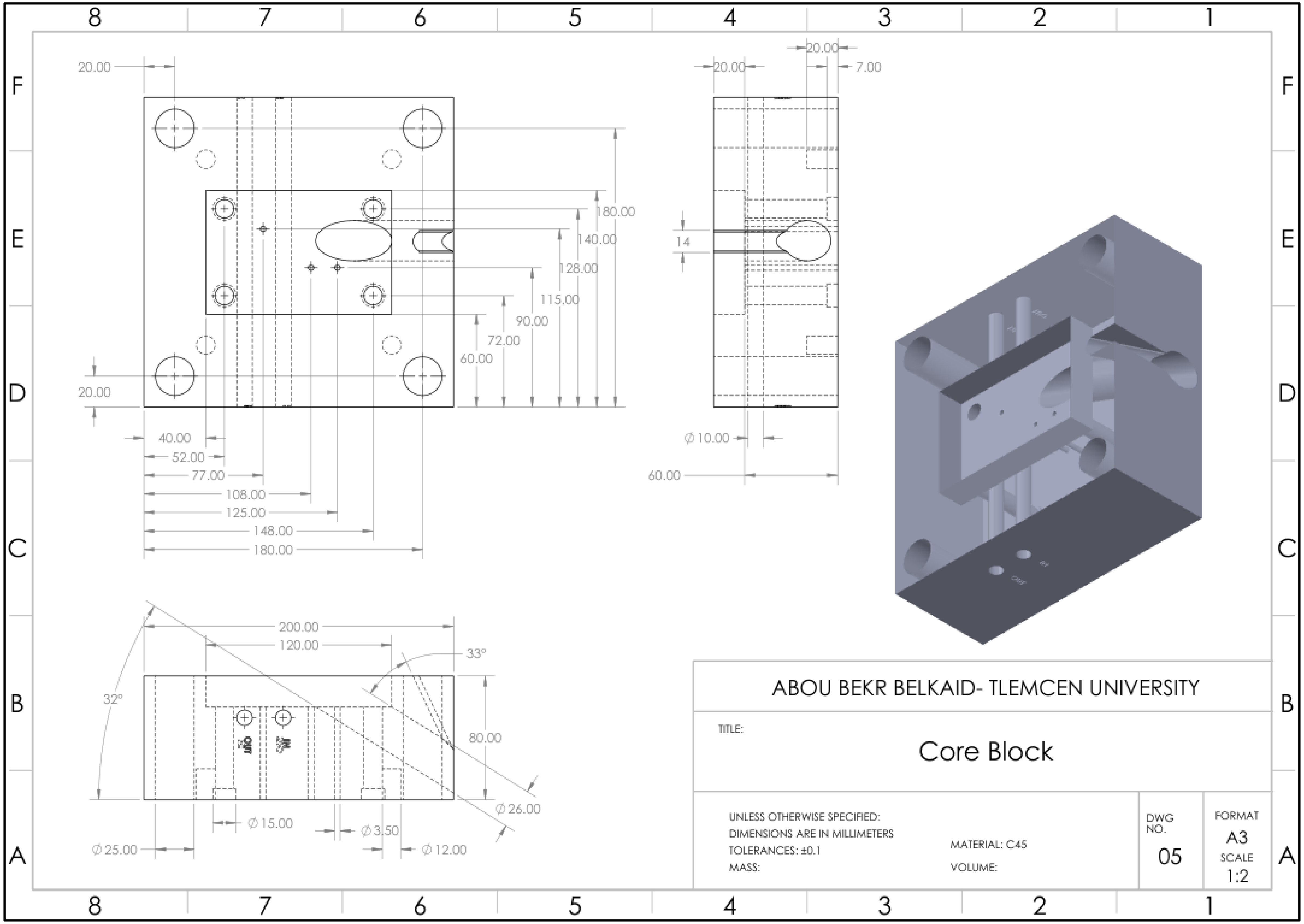


Fig.IV.10: core block

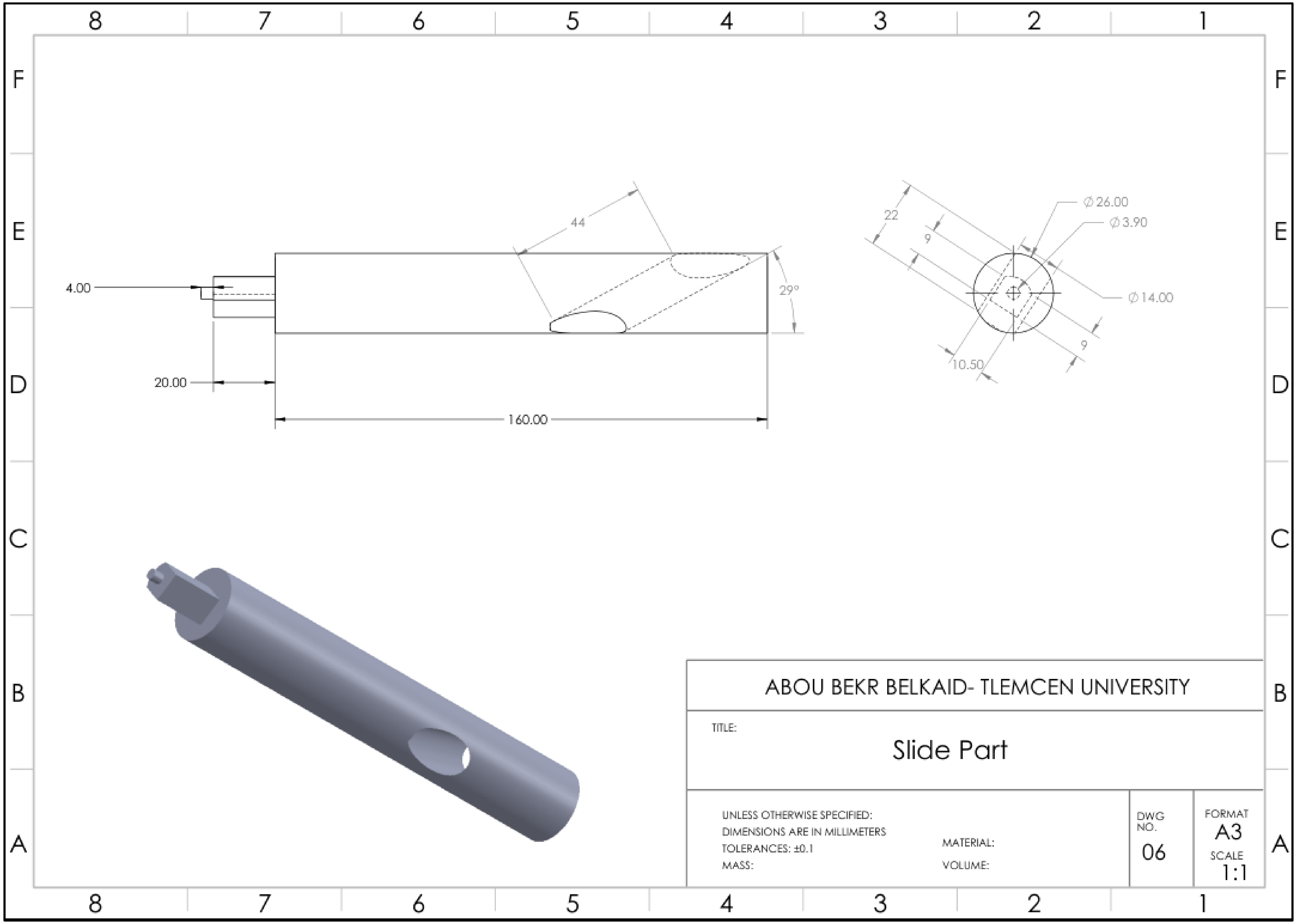


Fig.IV.11: slide part

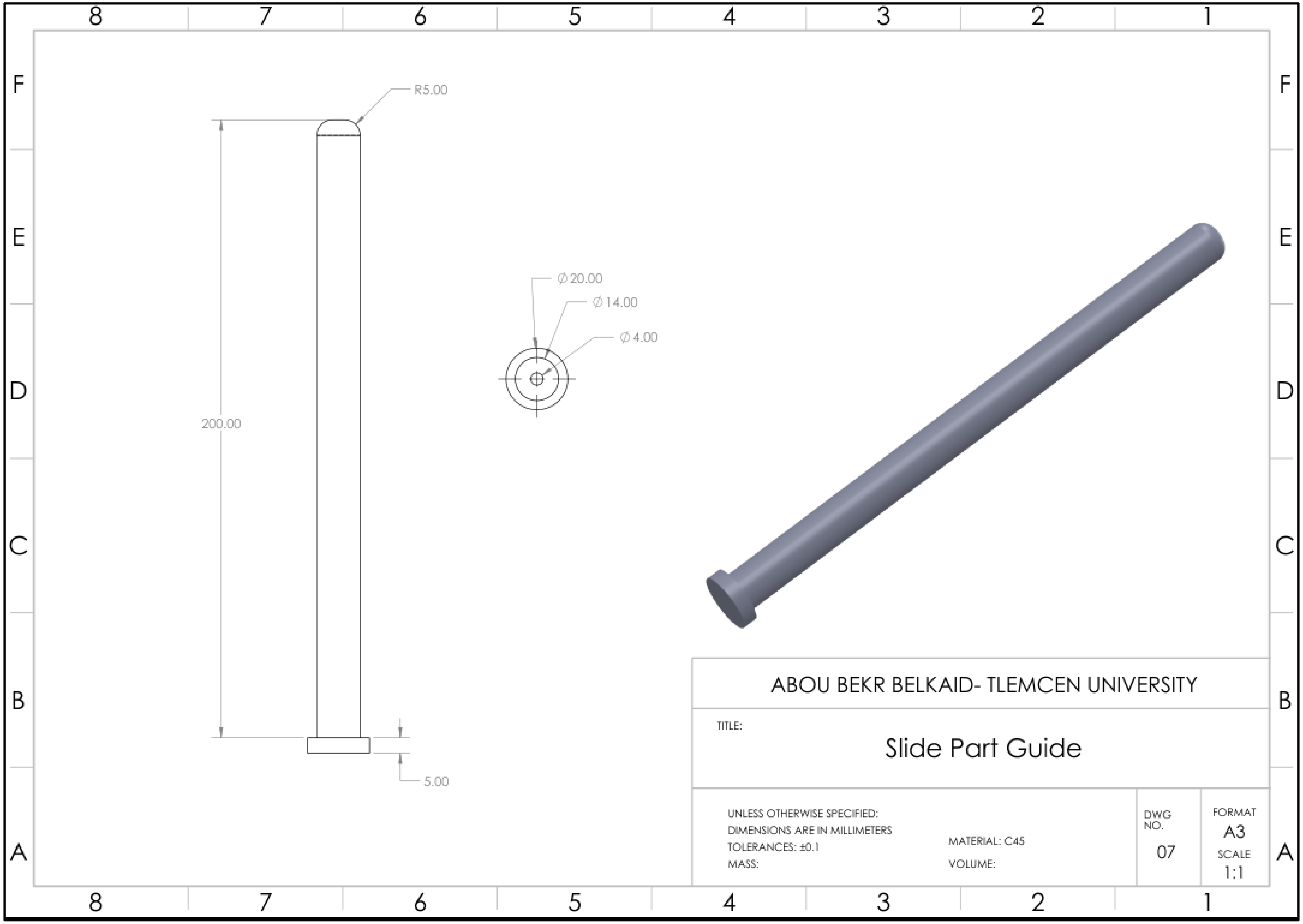


Fig.IV.12: slide part guide

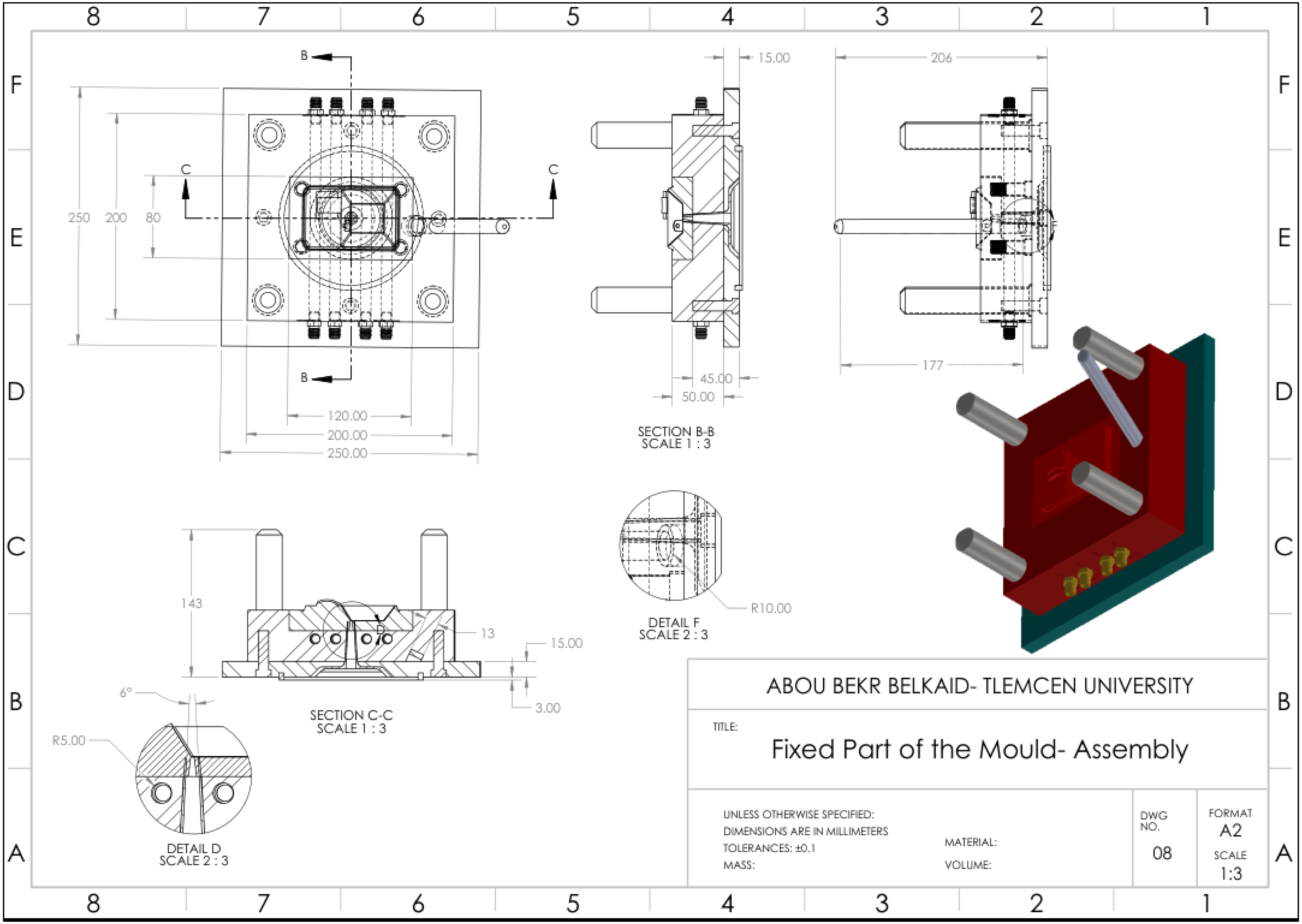


Fig.IV.13: Mould assembly- Fixed part

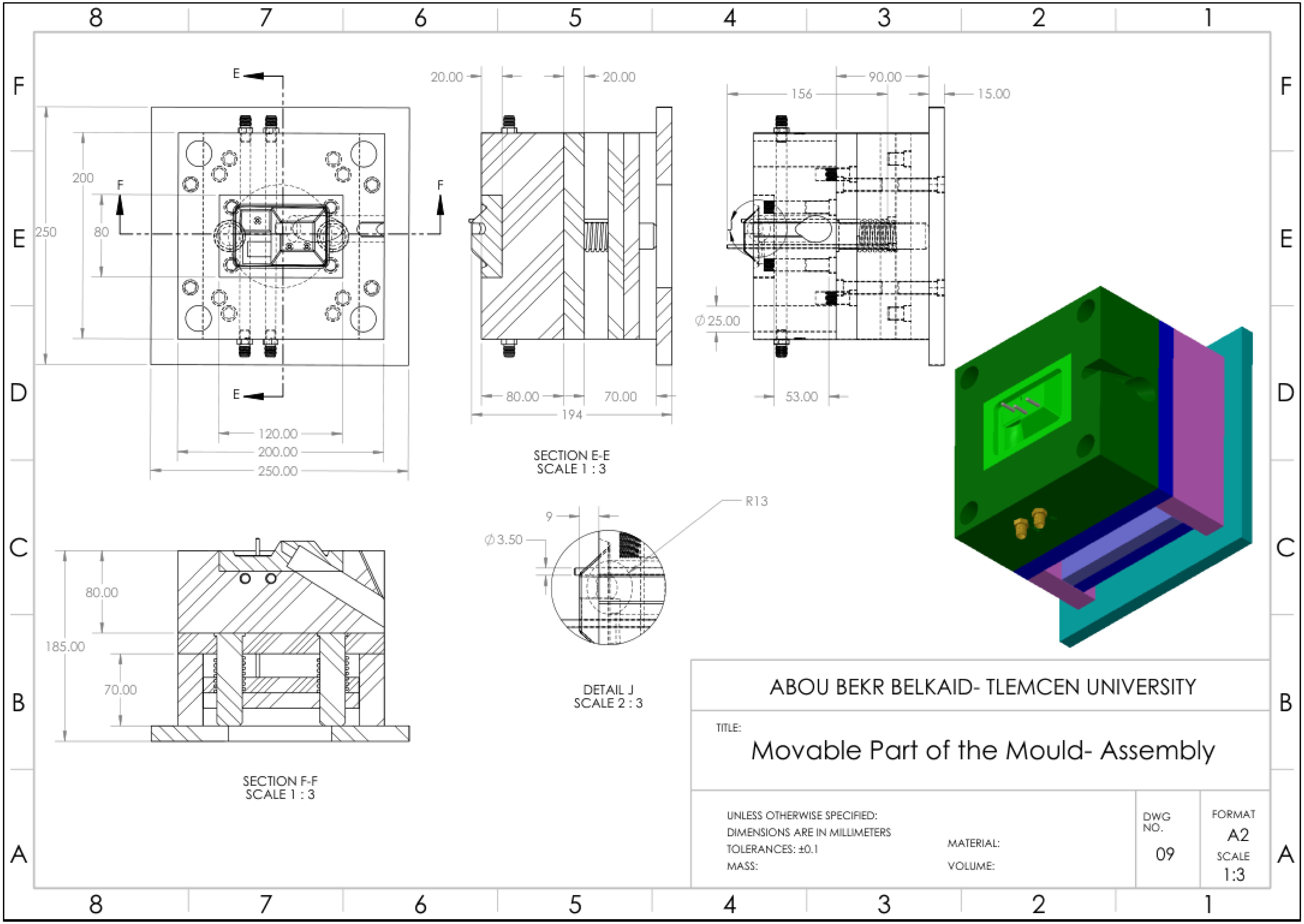
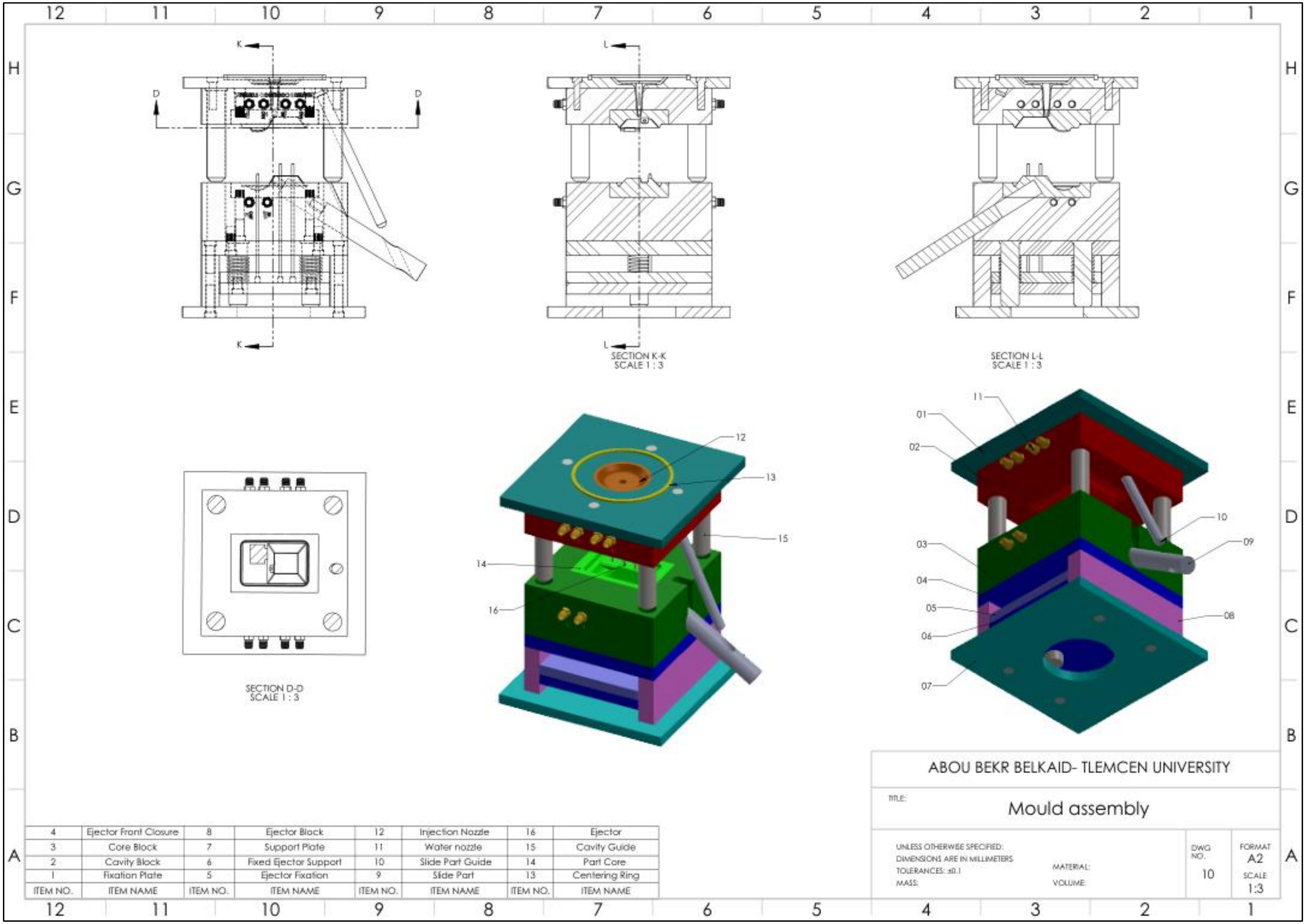


Fig.IV.14: Mould assembly- Movable part



ITEM NO.	ITEM NAME	ITEM NO.	ITEM NAME	ITEM NO.	ITEM NAME	ITEM NO.	ITEM NAME
4	Ejector Front Closure	8	Ejector Block	12	Injection Nozzle	16	Ejector
3	Core Block	7	Support Plate	11	Water nozzle	15	Cavity Guide
2	Cavity Block	6	Fixed Ejector Support	10	Slide Part Guide	14	Part Core
1	Fixation Plate	5	Ejector Fixation	9	Slide Part	13	Centering Ring

Fig.IV.15: Mould assembly

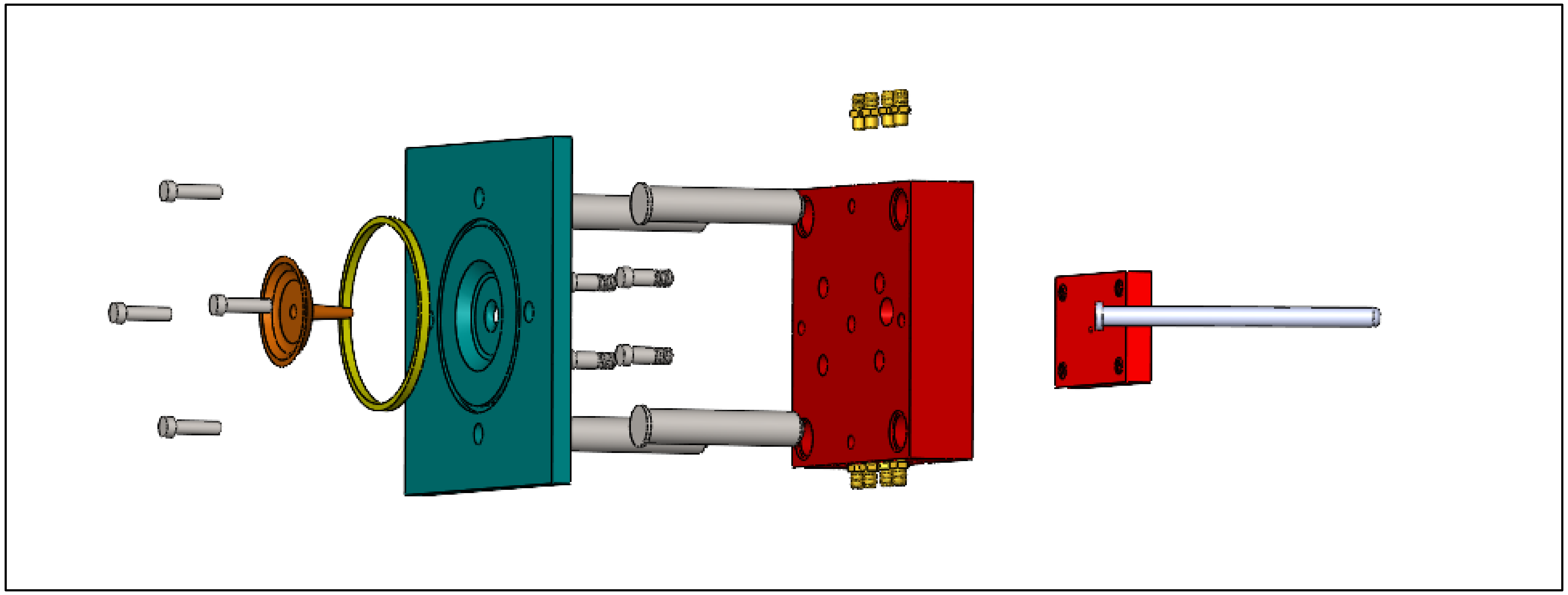


Fig.IV.16: Fixed part of the mould- exploded view

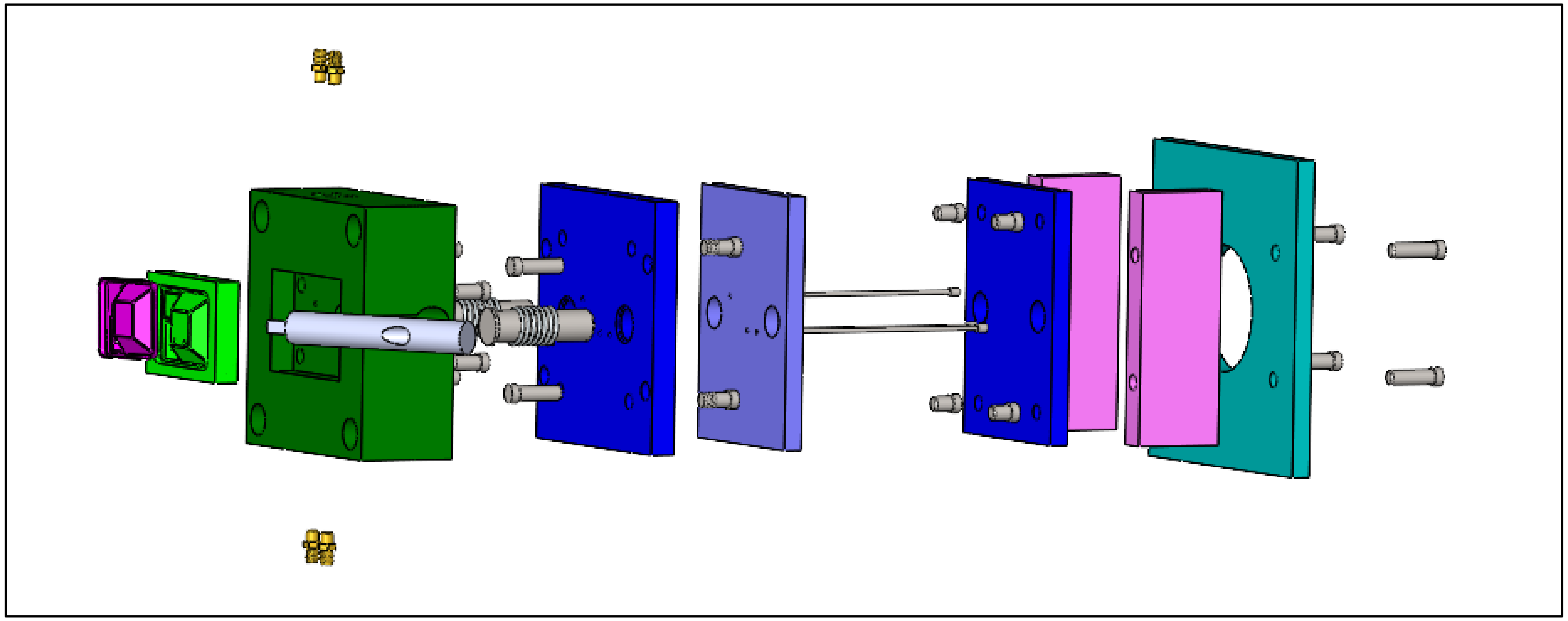


Fig.IV.17: Movable part of the mould- exploded view

4.8 Conclusion

In conclusion, Chapter 4 has focused on the crucial aspects of mould design for the car door handle cover in the context of enhancing the integration rate of the automotive sector. Mould design plays a pivotal role in the overall manufacturing process, and a well-executed design can significantly impact the quality, efficiency, and cost-effectiveness of production.

Throughout this chapter, we have explored the essential components and considerations involved in mould design, including the core block and cavity block design, cooling systems, and the movable parts of the mould. By delving into these topics, we have emphasized the importance of meticulous design and its contribution to achieving optimal outcomes in automotive manufacturing.

The knowledge and insights gained from this research will serve as a valuable resource for designers, manufacturers, and policymakers in the automotive industry. By leveraging the principles and techniques presented in this chapter, professionals can enhance their understanding of mould design and apply it to other components and products within the automotive sector.

Additionally, this chapter sets the stage for further exploration and analysis of mould design principles and advancements. As the automotive industry continues to evolve, it is essential to stay abreast of emerging technologies, materials, and methodologies in mould design. This continuous pursuit of innovation and improvement will drive the integration rate of the automotive sector, fostering collaboration, efficiency, and enhanced product quality.

GENERAL CONCLUSION

GENERAL CONCLUSION

This research project aimed to explore and address various aspects of the automotive manufacturing process, specifically focusing on the design and production of a car door handle cover through injection moulding. Throughout the study, we delved into the injection moulding process, part design considerations, simulation analysis, and mould design principles.

The investigation began with an exploration of the injection moulding process, highlighting the key components and operations involved. Understanding the functioning of injection machines and moulds was crucial in grasping the overall process.

Part design played a significant role in ensuring the functionality and durability of the car door handle cover. By following design rules, considering specifications, and verifying the design through various analyses, we aimed to create an optimal part design using SW 2022.

Simulation analysis provided valuable insights into the performance and behaviour of the designed part. Techniques such as thickness analysis, draft analysis, parting line and moulding direction analysis, and feeding system design helped evaluate and optimize the design for manufacturing.

Mould design was a critical aspect to ensure the accurate replication of the part. The creation of the core and cavity, footprints, and 2D drawings allowed for precise mould production.

However, due to time constraints and limitations in accessing the required materials and resources, the practical realization of the project could not be accomplished. Despite this limitation, the theoretical exploration and analysis conducted in this study have provided valuable knowledge and insights into the automotive manufacturing process, specifically in the context of injection moulding.

It is worth emphasizing that further research and practical implementation are required to fully realize the potential benefits and impact of this study. The lack of time and necessary materials hindered the actual production and testing of the car door handle cover. However, the findings and methodologies presented in this research can serve as a foundation for future endeavours in the field of automotive manufacturing.

Overall, this study shed light on the intricate processes and considerations involved in designing and manufacturing plastic parts through injection moulding. It highlighted the importance of part design, simulation analysis, and mould design in achieving functional and high-quality automotive components.

REFERENCES

REFERENCES

BIBLIOGRAPHIC REFERENCES

[1]: Lee, C. H., & Kim, S. (2015). Injection moulding handbook. Springer.

[2]: https://www.roechling.com/medical/technologies/injection-molding

[3]: Brydson, J. A. (2017). Plastics materials. Butterworth-Heinemann.

[4]: https://www.custompartnet.com/wu/InjectionMolding

[5]: https://img.directindustry.com/images_di/photo-mg/181922-14395441.webp

[6]: https://www.vnashengineering.com/images/products/EngineeringSolution/c2.png

[7]: Smith, J., & Johnson, A. (2021). Design Guidelines for Injection Moulded Parts.

[8]: Jones, R. T. (2020). Material Specifications for Injection Moulded Parts.

[9]: SolidWorks 2022 software (https://www.solidworks.com)

[10]: Kazmer, D. (2013). Injection Mold Design Engineering. Hanser Publications.

[11] : https://m.media-amazon.com/images/I/61lzrTXc57L._AC_UF894,1000_QL80_.jpg

[12] : chat.openai.com