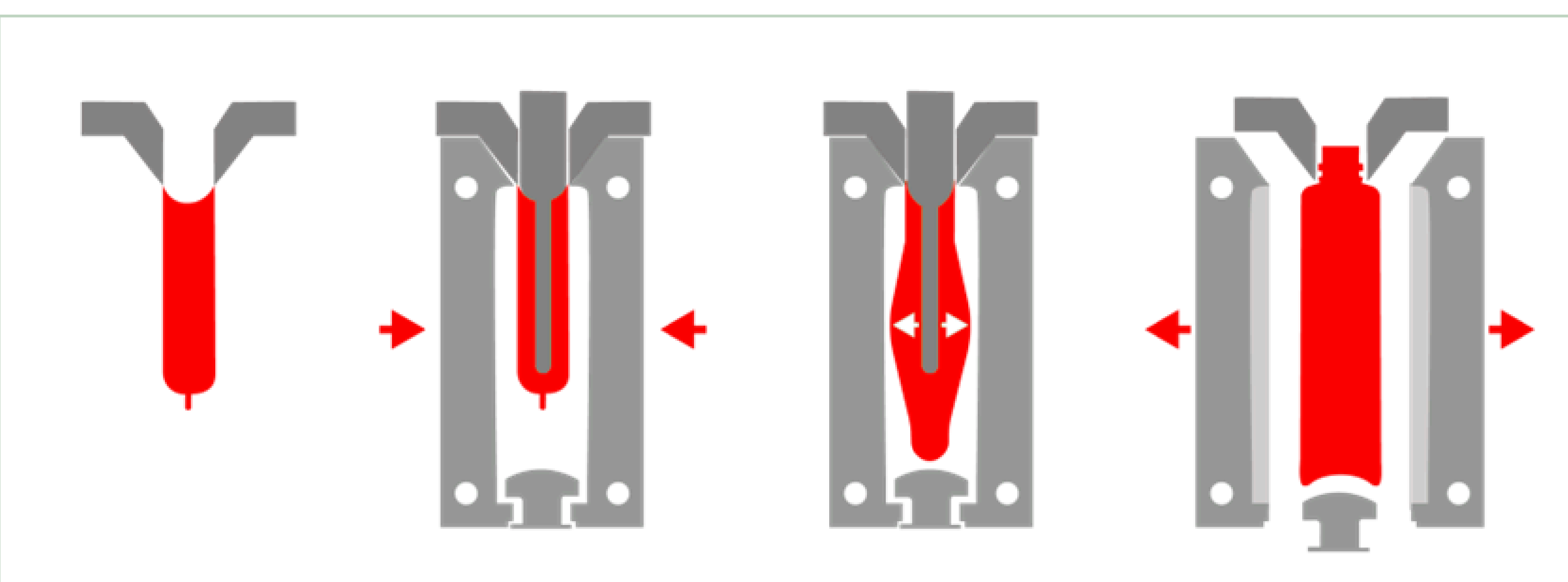
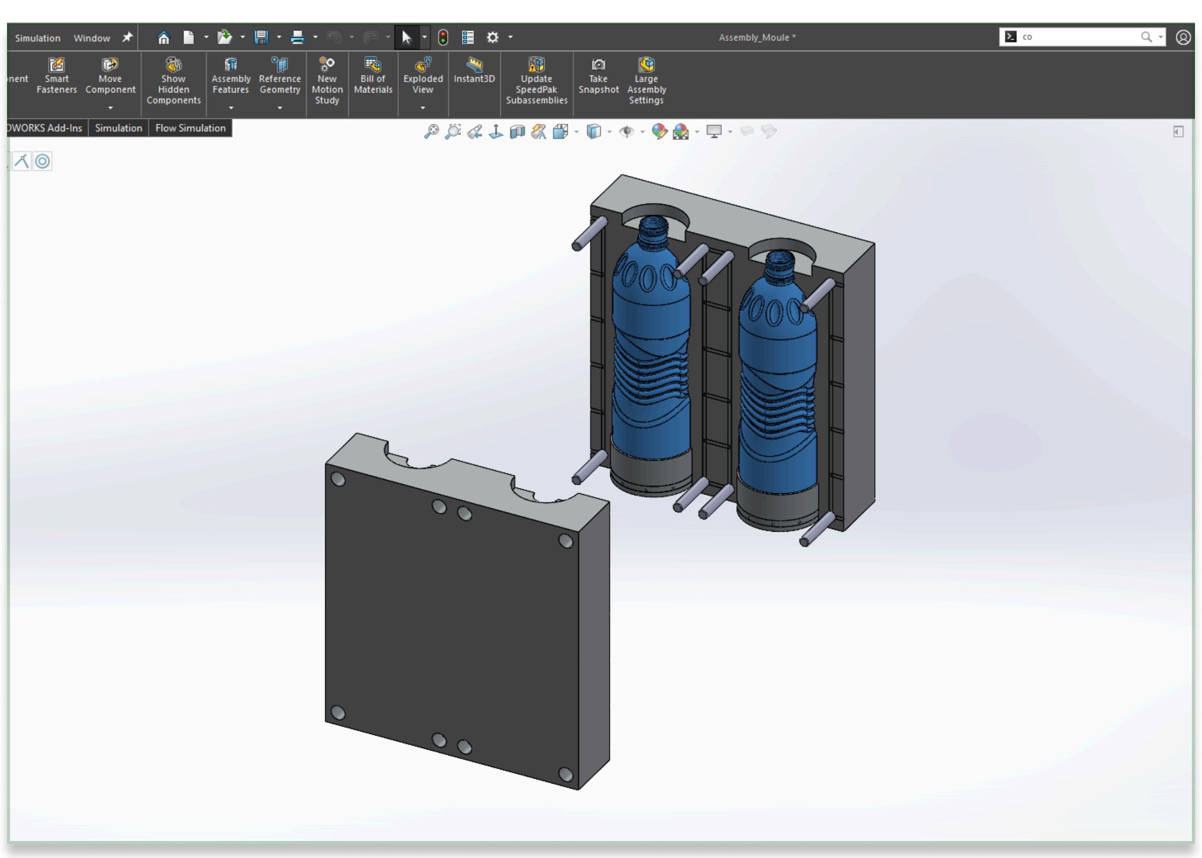
CASE STUDY

Conceptual Study of a Plastic Bottle and Its Mold

CAD Modeling Using SolidWorks





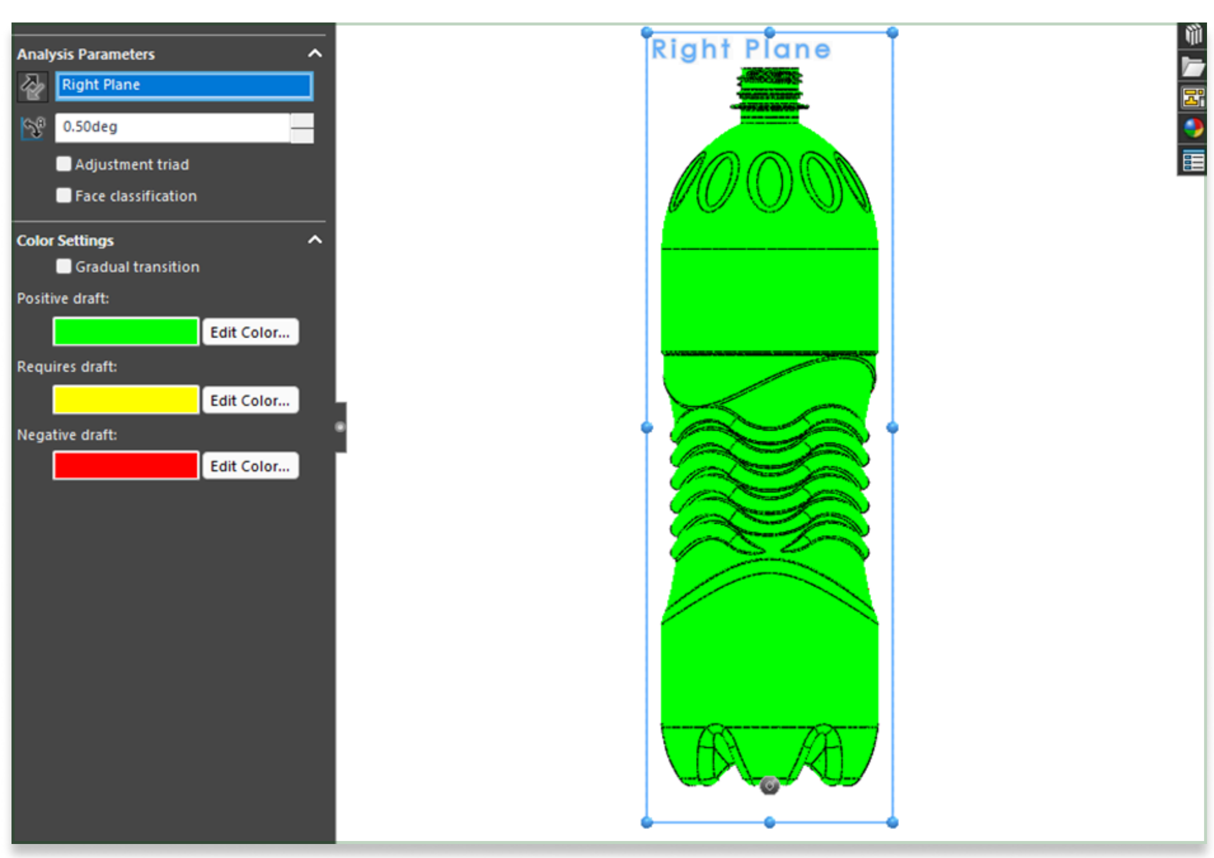


Table of Contents

Abstract

This study explores the challenges associated with designing a **plastic bottle ready for molding**, highlighting the requirements of a Design for Manufacturing (DFM) approach. Using **SolidWorks** mold design tools, it addresses the integration of complex features and adaptation to blow molding constraints. The analysis emphasizes the importance of making adjustments to ensure mold feasibility and final product quality, while optimizing production efficiency.

Introduction

This study presents the design of a plastic bottle, an essential step in the development of the mold intended for its production. It highlights the need to reconcile aesthetics, ergonomics, and structural performance. The analysis focuses on creating a bottle with embossed grip patterns for optimal handling, an ergonomic shape, and a reinforced base for stability, key features that enhance brand identity and user comfort.

The realization of such complex geometries presents major challenges in both design and engineering, requiring precise integration of production constraints.

The use of advanced mold design tools in SolidWorks offers several key advantages for the design and manufacturing of molds :

- **Undercut Analysis** : Identifies and corrects areas that are difficult to demold, ensuring defect-free production.
- **Core and Cavity** : Facilitates the separation of the mold into two parts (fixed and moving) to guarantee efficient and precise molding.
- **Parting Line Creation** : Determines the optimal location of the parting line to ensure a tight mold closure and minimize visible marks on the final part.
- **Mold Cooling Issues** : Allows for the simulation and optimization of cooling channels to ensure uniform cooling, thereby reducing cycle times and shrinkage defects.

Introduction (02)

- **Shrinkage / Scale Factor** : Anticipates dimensional variations caused by plastic cooling by adjusting mold dimensions to achieve a final product that meets specifications.

Expertise in 3D modeling with SolidWorks, combined with in-depth mastery of mold design tools, makes it possible to successfully overcome these challenges. This enables the design of complex, mold-ready geometries that not only meet aesthetic and ergonomic requirements but are also optimized for efficient production.

Injection-Blow Molding Process in Cold Cycle

The manufacturing process of a plastic bottle is divided into three key stages, described as follows :

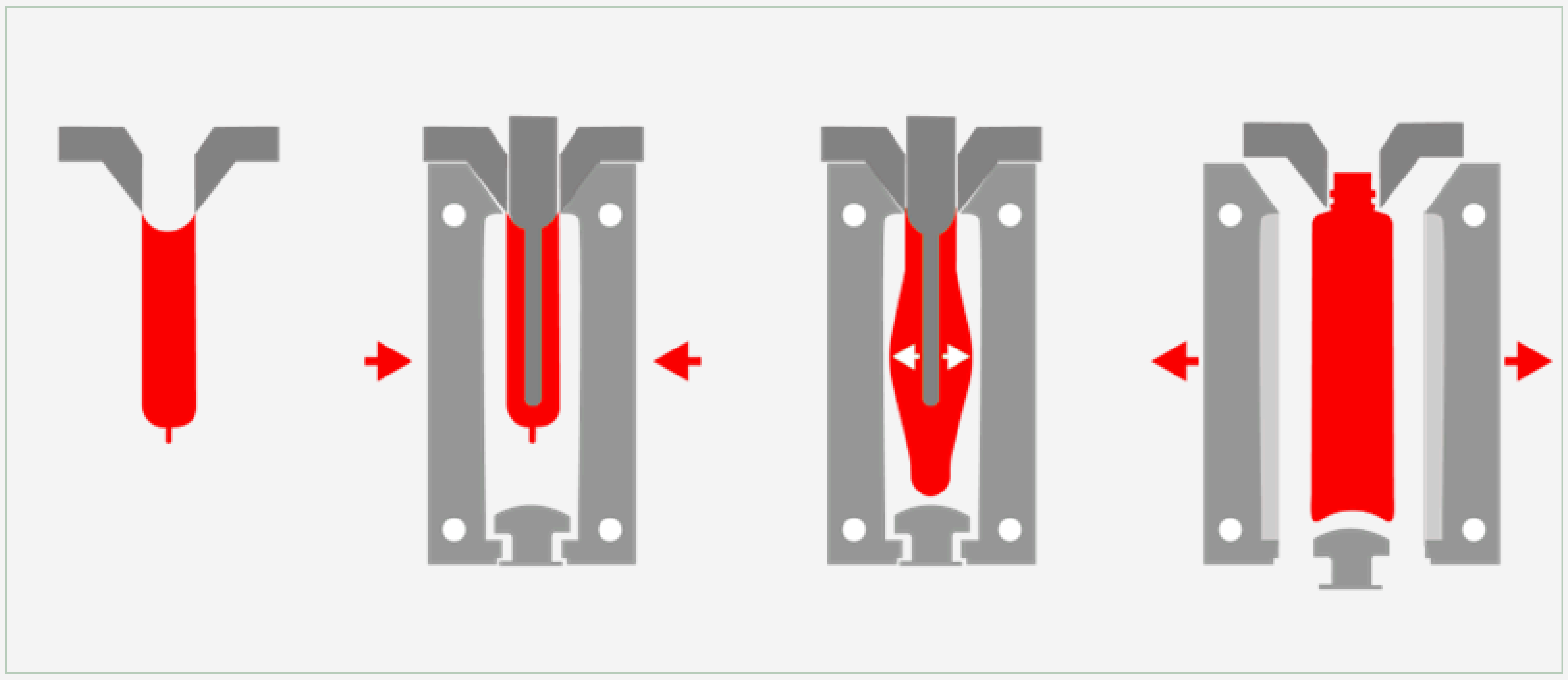
1. **Preform Manufacturing**
    - **Injection** : The base material, typically PET, is heated and injected into a mold to create a preform.
    - **Preform** : This is a small plastic tube that already contains the approximate shape of the final bottle and will serve as the raw material for the next stage.

2. Preform Blow Molding

- **Reheating** : The preform is reheated to a controlled temperature (around 100 °C) to make it malleable.
- **Blow Molding** : Placed inside a mold shaped like the final bottle, a stream of compressed air (between 25 and 40 bar) is injected to stretch the preform and give it its final shape.

3. Cooling and Finishing

- **Cooling** : Once the shape is formed, the mold is cooled through dedicated cooling channels, ensuring rapid and uniform cooling of the bottle.



Injection-blow molding process (Figure 1) source

CAD Modeling Using SolidWorks

1.5L Bottle Design

The modeling of the plastic bottle was carried out using a series of advanced features in SolidWorks, enabling the creation of a product that is both ergonomic and visually appealing, while also meeting manufacturing constraints.

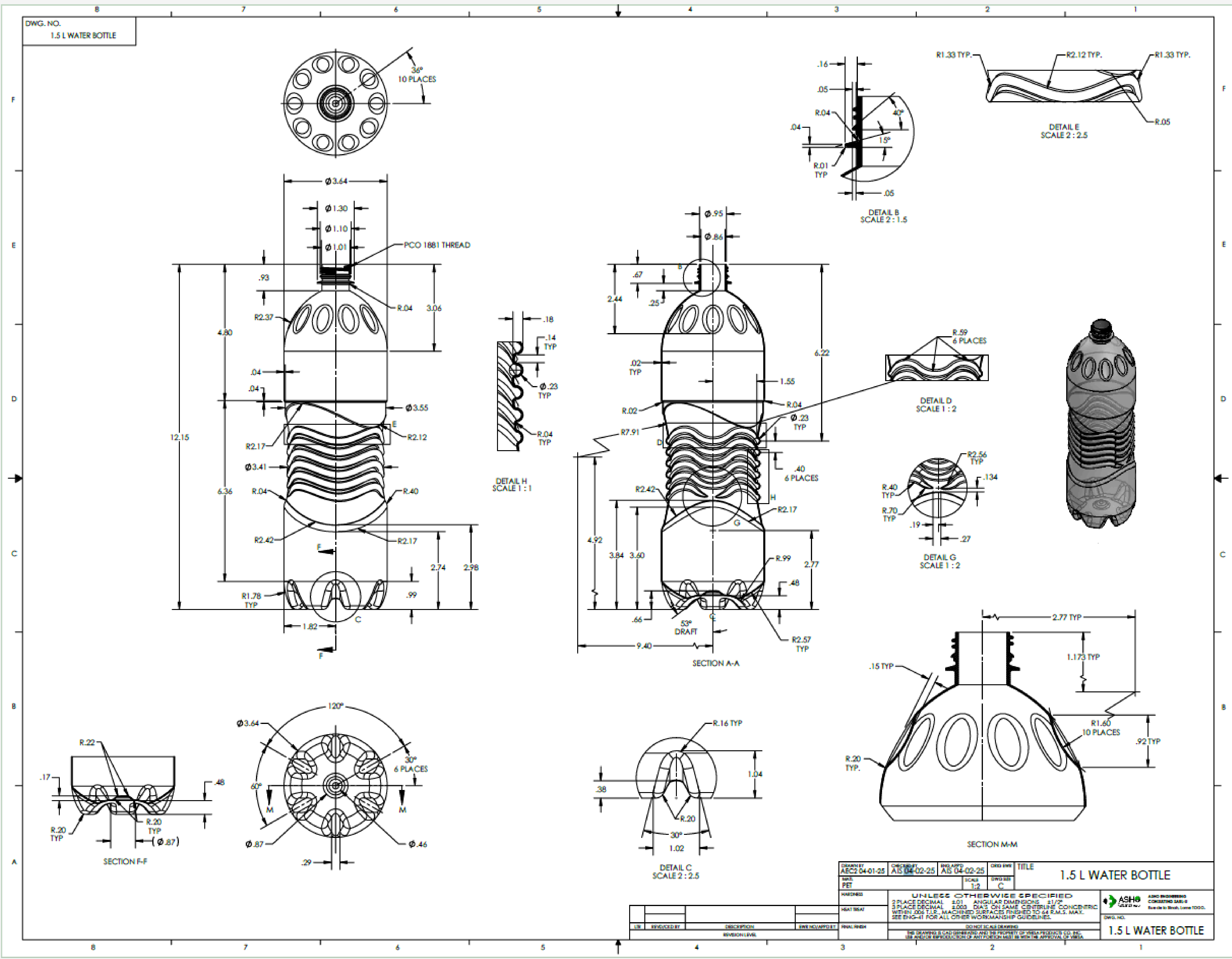
1. Creation of the Base Geometry

- The design begins with a Revolved Boss/Base, using a 2D profile rotated around a central axis to generate the main cylindrical shape of the bottle.
- This step provides a **symmetrical base geometry** consistent with the bottle's dimensions.

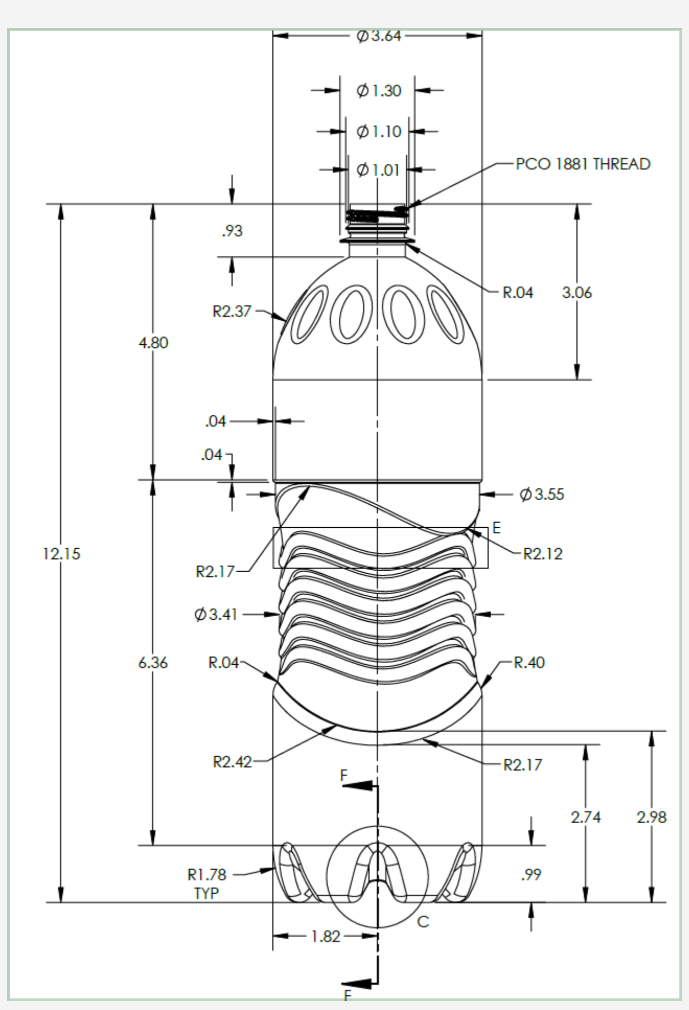
2. Shape Refinement and Functional Details

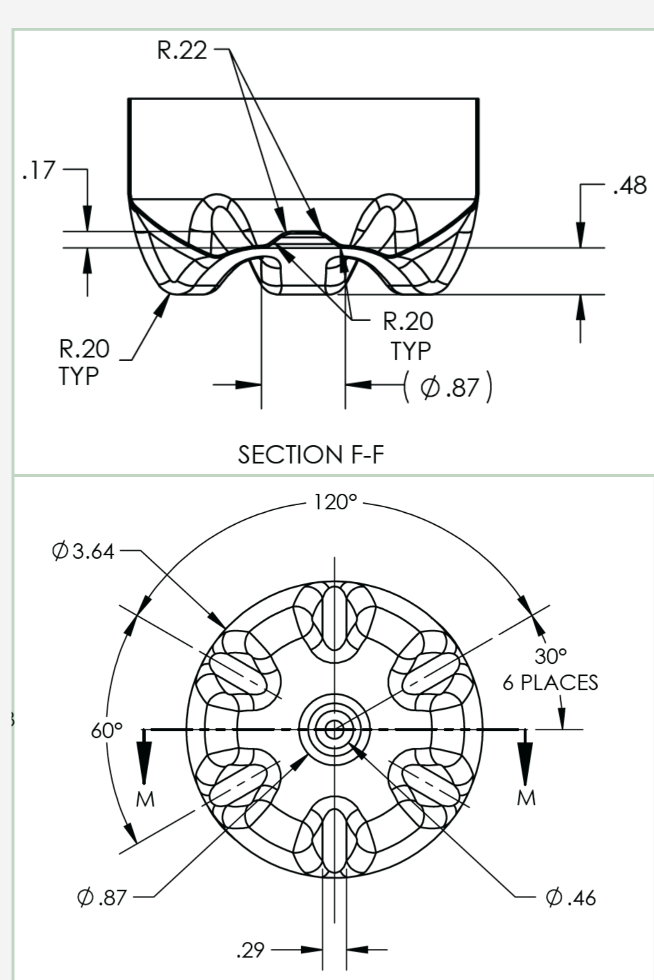
- An **Extruded Cut** was used to adjust certain areas of the bottle, such as the neck or the base.
- A **Swept Cut** was applied to create grooves or ergonomic features that enhance grip.
- A **Circular Pattern** was used to replicate repeating elements, such as the textures on the cap.
- A **Fillet** was applied to the neck of the bottle to enable proper threading for the cap.

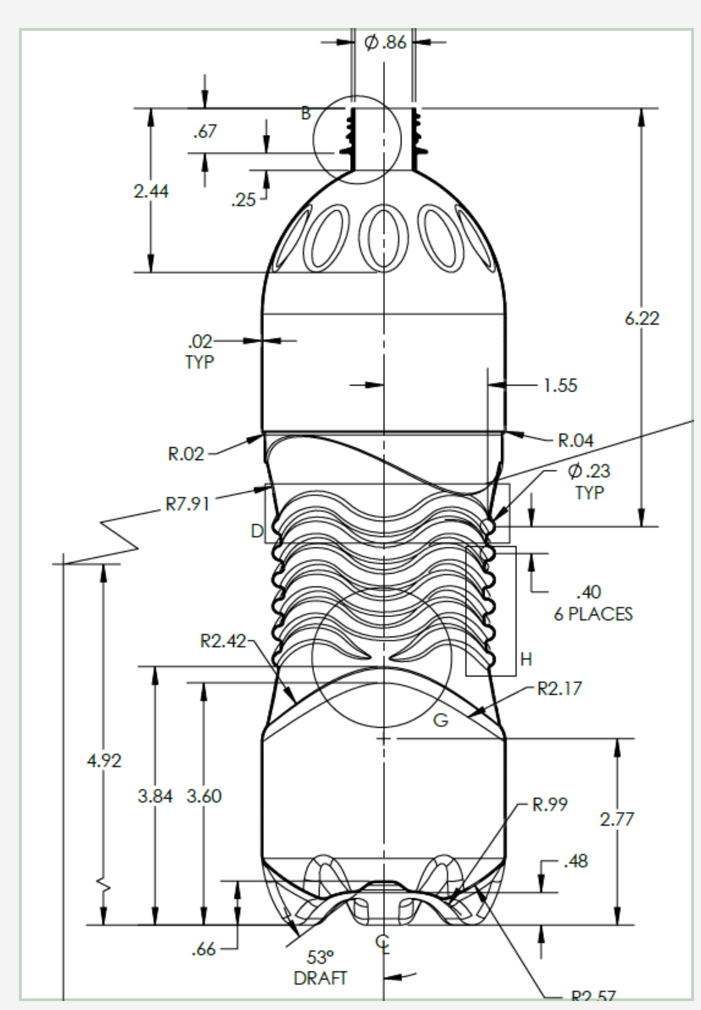
CAD Modeling Using SolidWorks (02)



***2D** Bottle drawing (Figure 2)*







CAD Modeling Using SolidWorks (03)

***3D** Bottle drawing (Figure 3)*

Mold Design

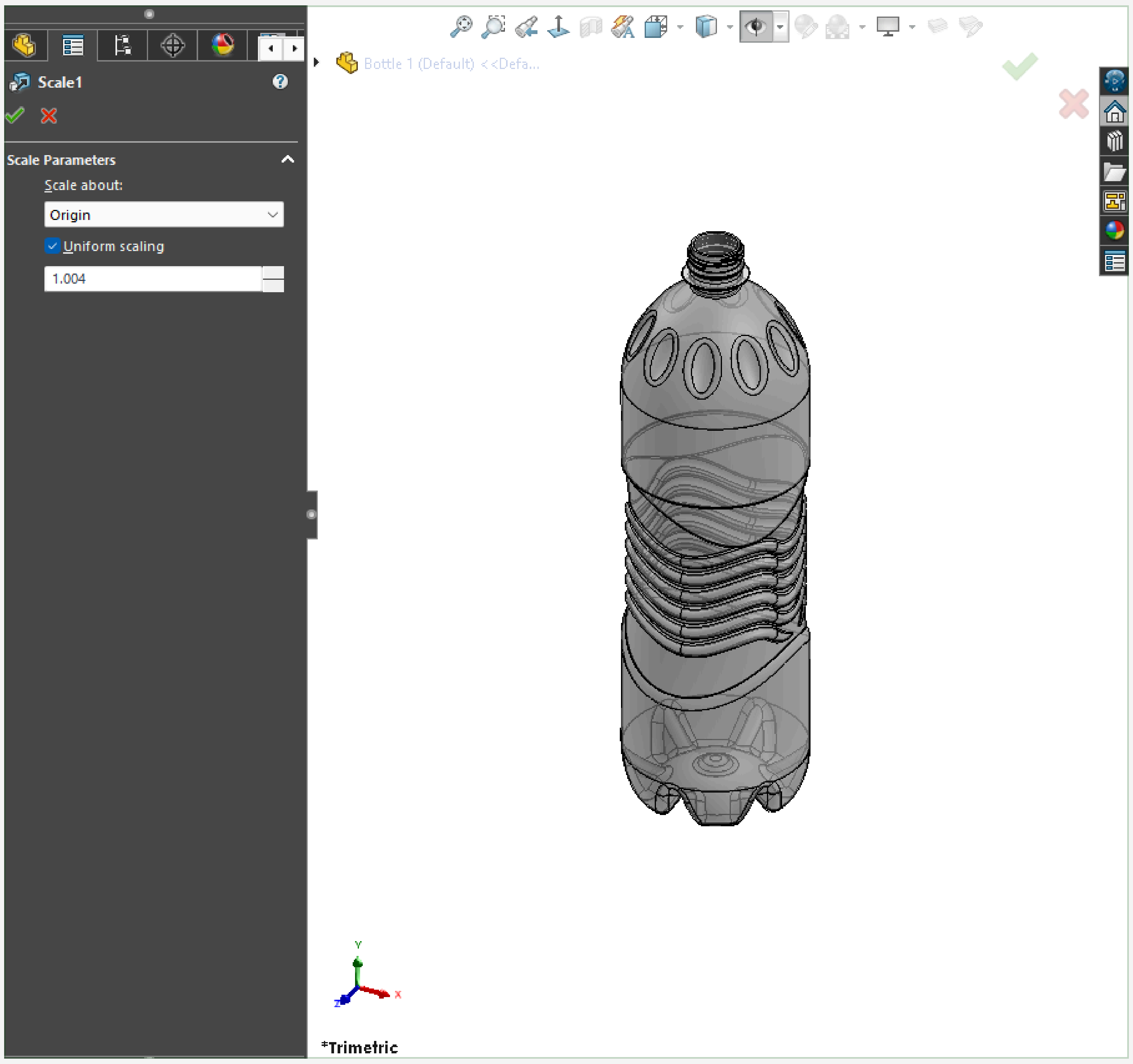
The mold design for the plastic bottle required the use of **SolidWorks' advanced mold tools** to ensure optimal manufacturability. The key steps in the process are outlined below :

1. Scale

Before starting the mold design, a scale factor was applied to the bottle to compensate for shrinkage during cooling and ensure the part returns to its intended shape.

In this case, a scale factor of 1.004 relative to the original model was used ; this is a commonly recommended value for polyethylene terephthalate (PET) plastics.

CAD Modeling Using SolidWorks (04)

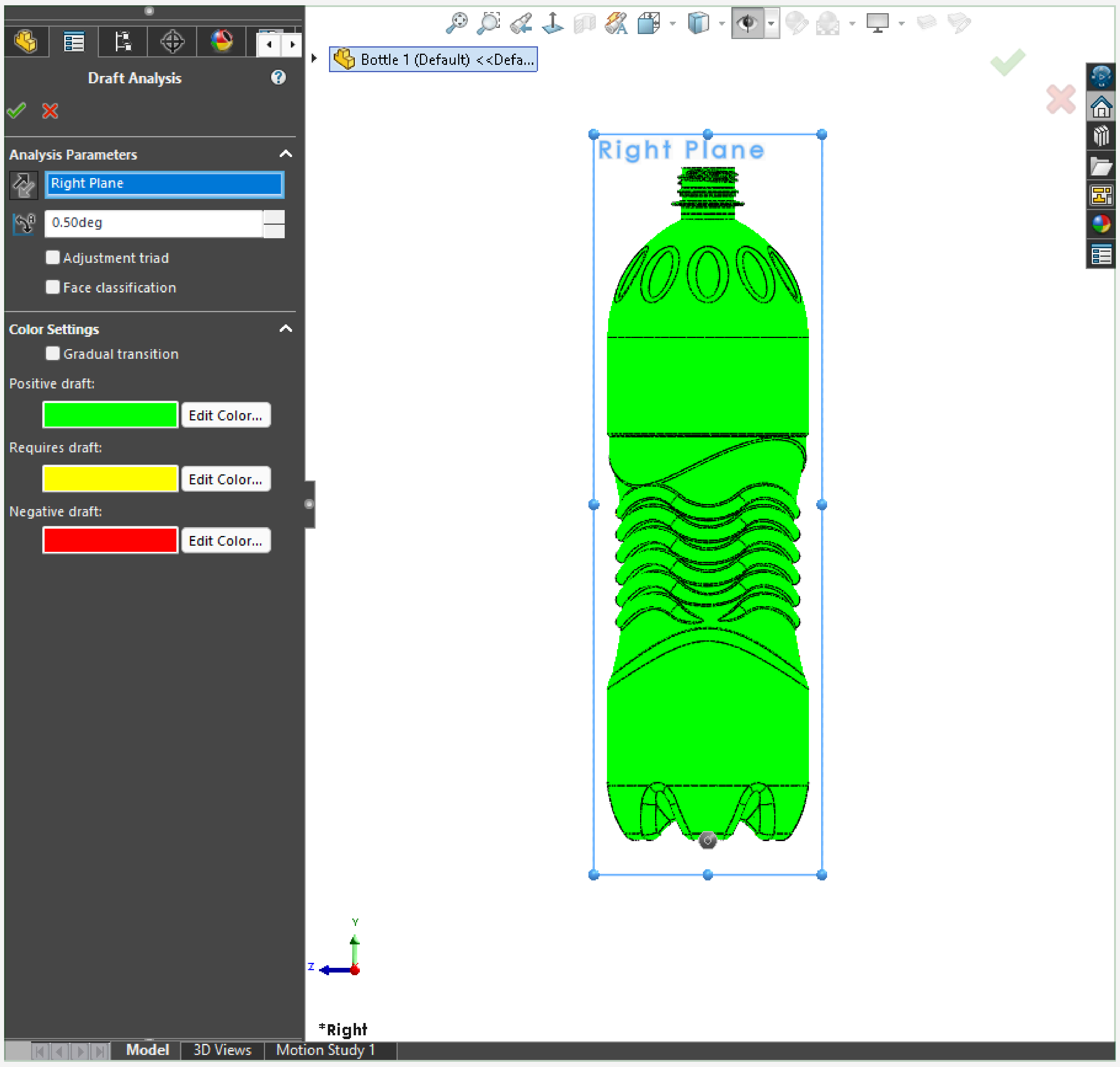


Bottle scaling (Figure 4)

2. Draft and Undercut Analysis

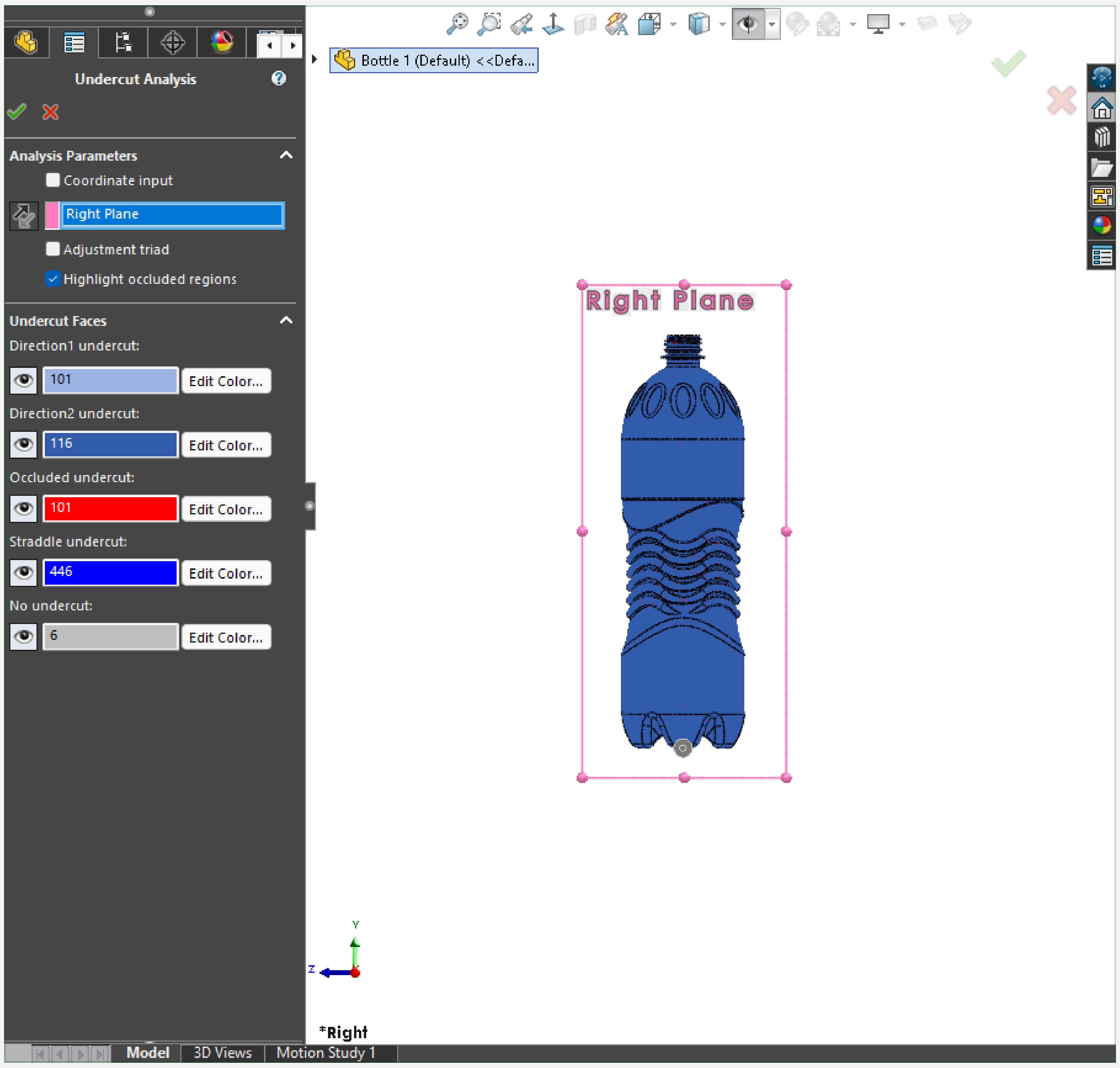
The draft angle analysis of the part ensures smooth mold release. SolidWorks' **Mold Tools** offers a dedicated feature to detect drafts and undercuts, making it easier to identify areas that require modification.

The required draft angle depends on the type of plastic used. For PET, a recommended draft angle of 0.5° is applied to ensure optimal demolding. The image below shows that with a 0.5° draft, the angle remains positive, allowing for easy part ejection from the mold.



Draft analysis (Figure 5)

The undercut analysis helps identify areas that require the addition of cavities. In this case, the integration of a second cavity is essential to accurately define the lower portion of the bottle.



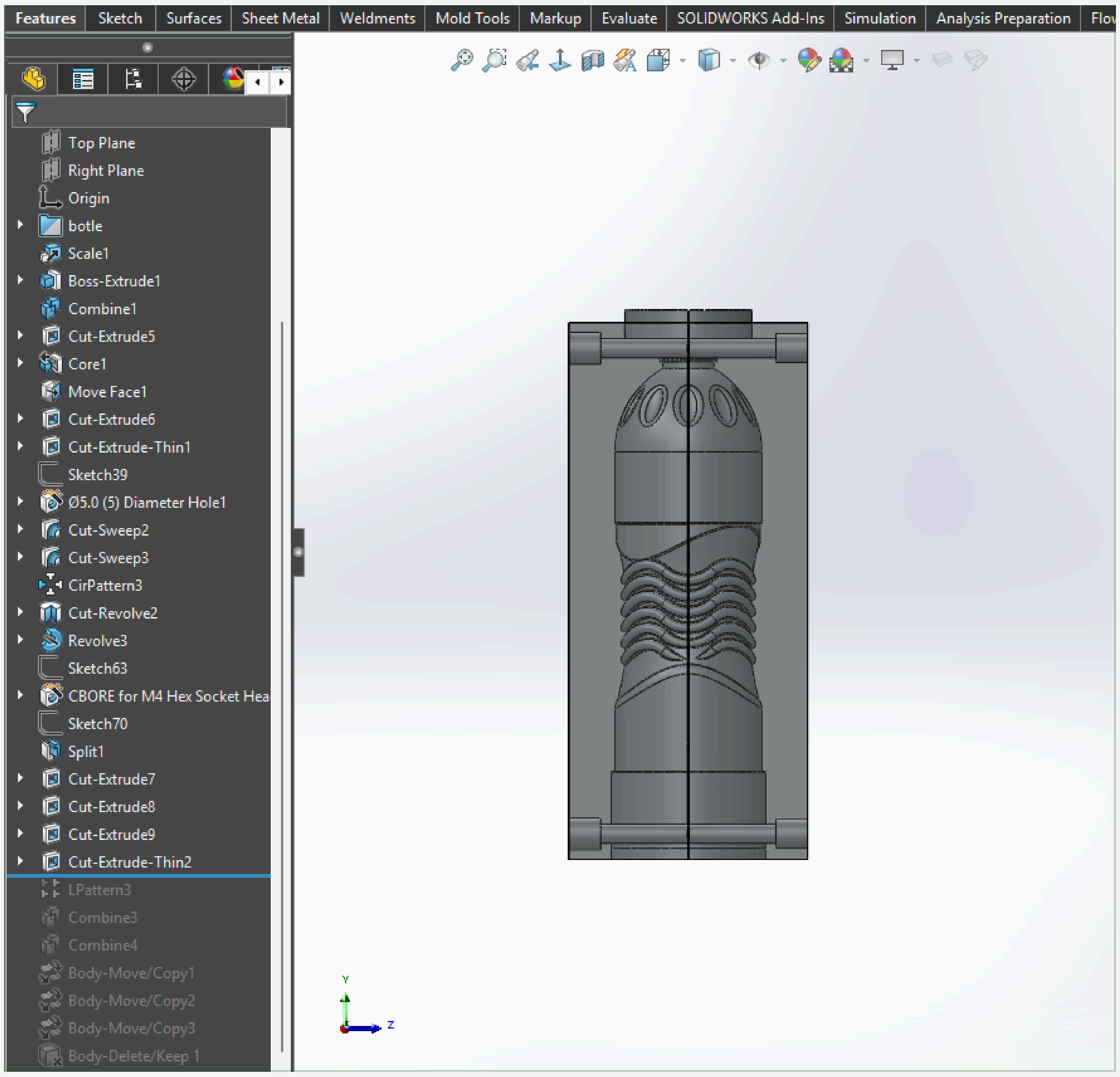
Undercut analysis (Figure 6)

3. <u>Parting Line Analysis and Mold Separation</u>

The parting line was defined based on the geometry of the bottle. Its placement was guided by several key criteria:

- Minimizing visible parting defects on the bottle after molding.
- Ensuring even distribution of plastic material within the cavity.
- Facilitating mold opening without altering the bottle’s shape.

CAD Modeling Using SolidWorks (07)



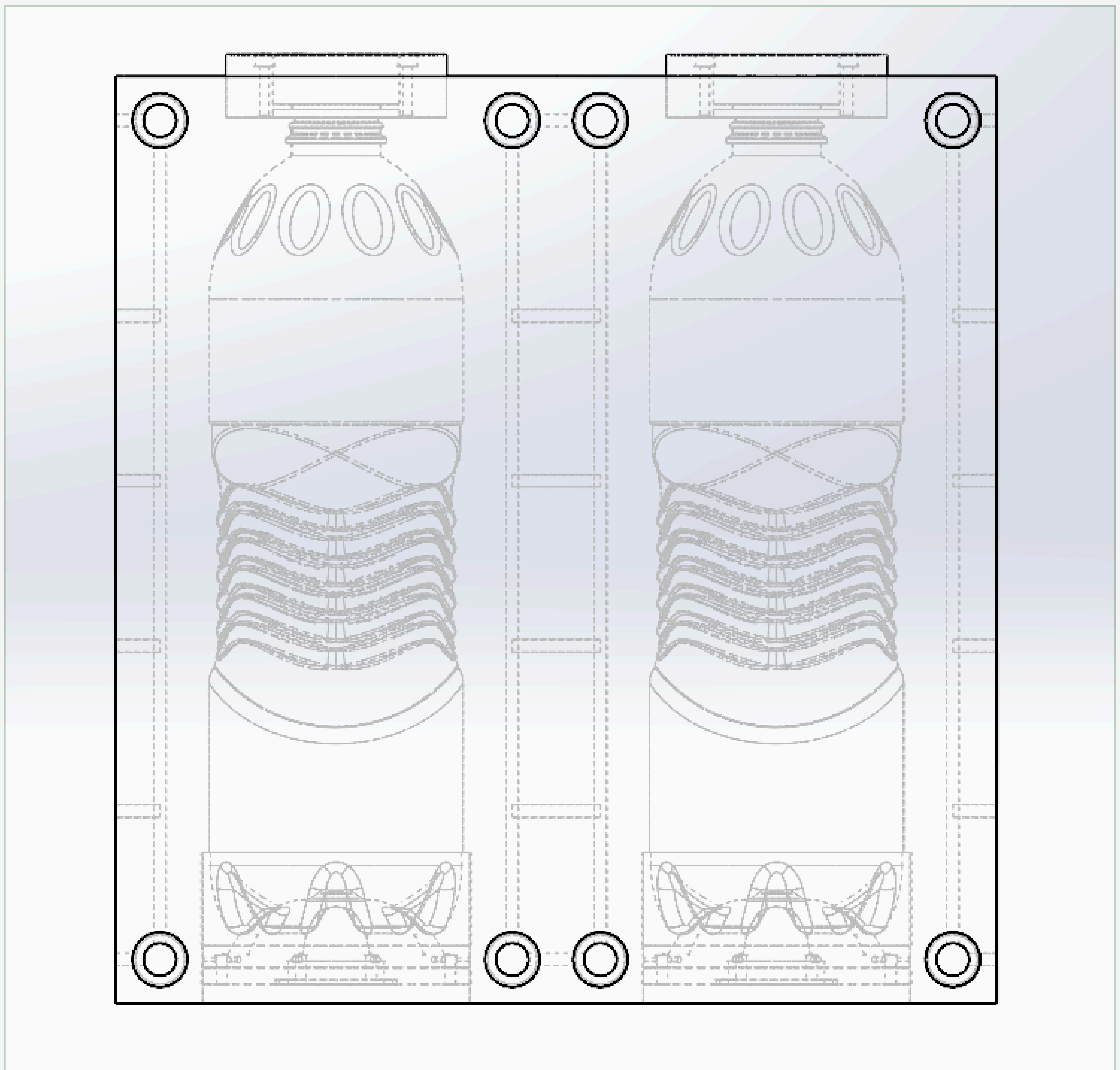
Mold separation (Figure 7)

4. Ejection Ease and Deformation Control

The ejection of the bottle is optimized through several elements :

- **Draft Angles** : Sufficient draft angles were applied to vertical surfaces to prevent the bottle from sticking in the mold.
- **Optimized Cooling** : Cooling channels were incorporated to regulate mold temperature and prevent thermal deformation—particularly at the bottom of the bottle, which is more prone to dimensional changes.

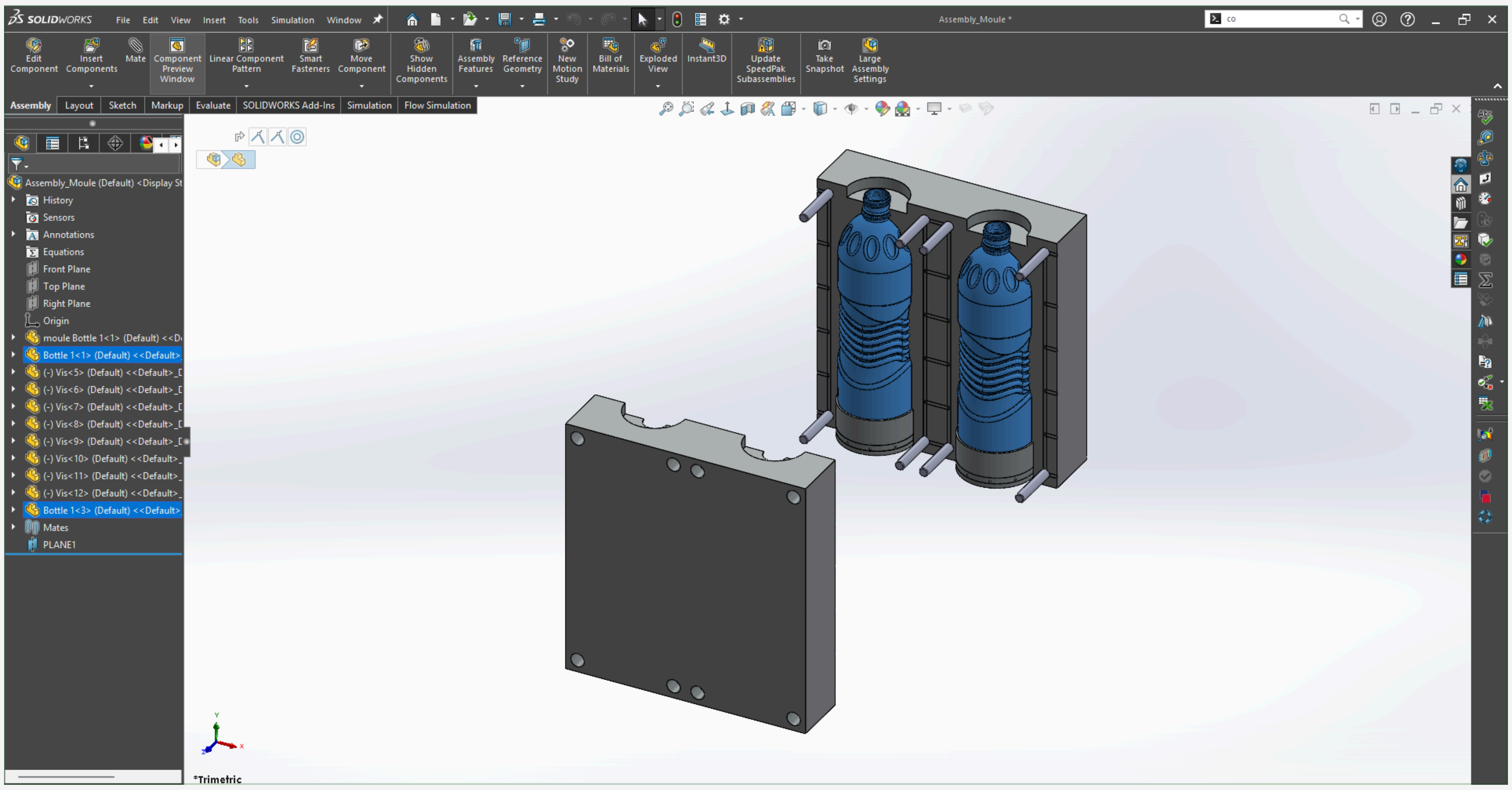
CAD Modeling Using SolidWorks (08)



Mold cooling channels (Figure 8)

This rigorous design process ensures smooth and efficient manufacturing while maintaining high production quality.

CAD Modeling Using SolidWorks (09)



3D Modeling of the 1.5L double-cavity mold (Figure 9)

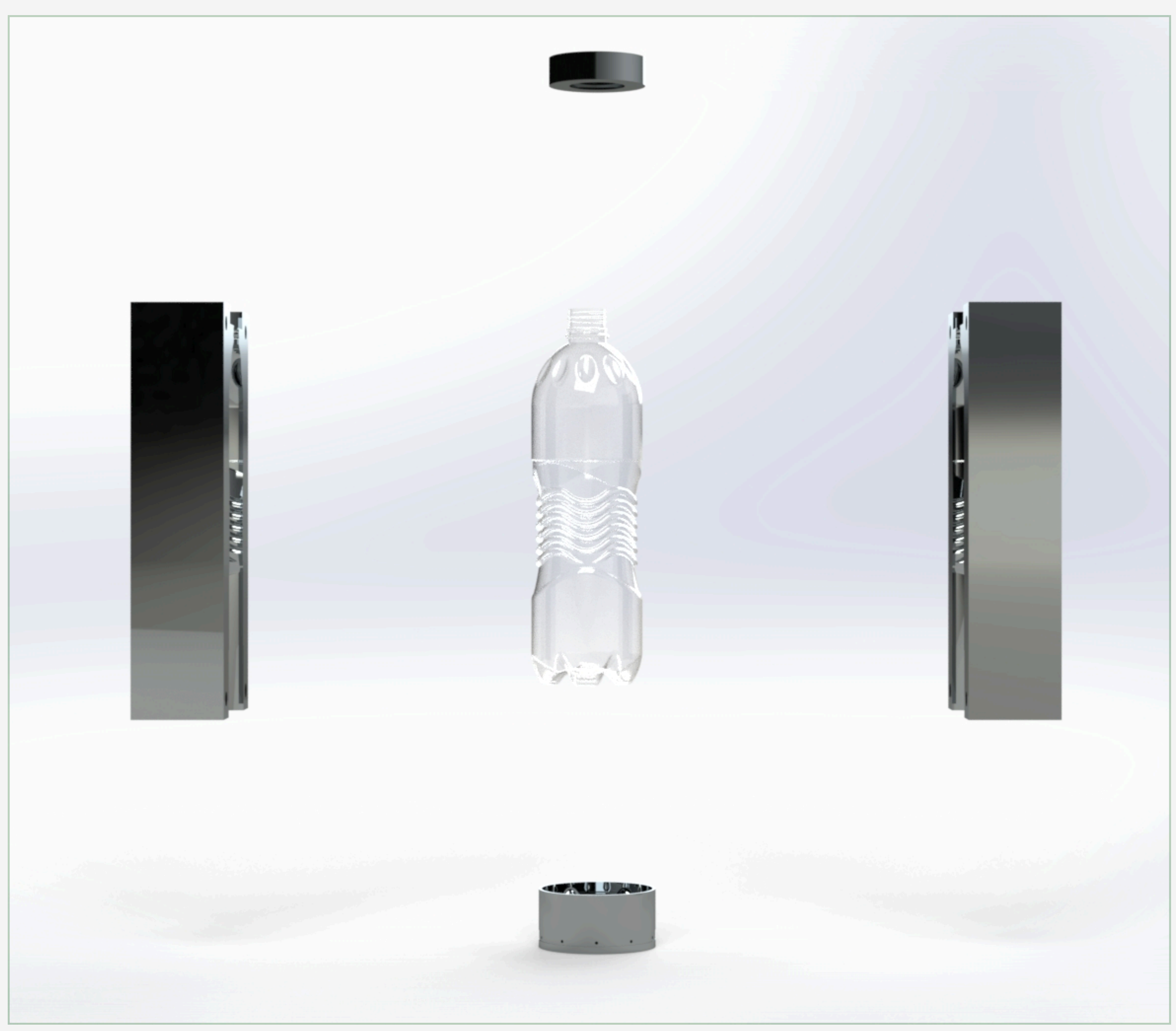
Following the mold design, it becomes clear that the quality of a mold goes beyond its internal dimensions. The tool's surface finish and the precision of mold closure also play a critical role in the performance and reliability of production.

These factors combined with accurate modeling, ensure that the mold meets both technical requirements and industry quality standards.

CAD Modeling Using SolidWorks (10)

***3D** Design of the bottle and its mold* (*Figure 10.1*)

***3D** Design of the bottle and its mold* (*Figure 10.2*)



Conclusion

The conceptual study of this plastic bottle and its mold highlights the importance of integrating manufacturing constraints from the earliest stages of modeling. By relying on the injection-blow molding process and **SolidWorks**' advanced tools, it is possible to develop complex shapes that meet aesthetic, ergonomic, and functional requirements ; while remaining compatible with efficient industrial production.

The integration of key parameters such as shrinkage factor, draft angles, and cooling circuits helps optimize final product quality, ensure mold feasibility, and reduce cycle times. This approach thus facilitates a smooth transition from design to manufacturing, while minimizing last-minute adjustments.

By ensuring rigorous and technically consistent modeling, it becomes a strategic lever for the development of complex plastic components that meet industry standards ; combining performance with manufacturing efficiency.

Feel free to contact us at info@asho-engineering for more information.

*At Asho Engineering, our **certified CSWP** engineers are available to assist you with your **design** and **engineering** projects. We offer high-quality services tailored to the needs of SMEs and startups, with innovative and competitive solutions. Beyond this study, our expertise covers a wide range of CAD and product engineering applications. So, how about **we discuss** your next project?*

Explore similar content

- ***ASHO Engineering: Case Studies***
- ***ASHO Engineering: Blog***