MEM 687: Manufacturing Process I

Manufacturing Process I - Project

**Saurabh Pethkar - **(sp3838@drexel.edu)

Instructor – Dr. Gongyao Zhou

Table of Contents

Introduction

Injection molding is a manufacturing process widely used for producing parts by injecting molten material into a mold. This process can be performed with a variety of materials, including metals (known as die-casting), glasses, elastomers, confections, and most commonly, thermoplastic, and thermosetting polymers. The essence of injection molding revolves around its versatility and efficiency, making it an indispensable technique for the mass production of intricate and detailed components that are used across numerous industries, from automotive and consumer electronics to medical devices and packaging.

The process begins with the melting of plastic pellets in the injection molding machine's barrel, where heat and mechanical energy are applied. The molten plastic is then forced into a mold cavity that defines the shape and size of the final part. After the material is injected, it cools and solidifies to form the desired part. The mold then opens, and the part is ejected, completing the cycle.

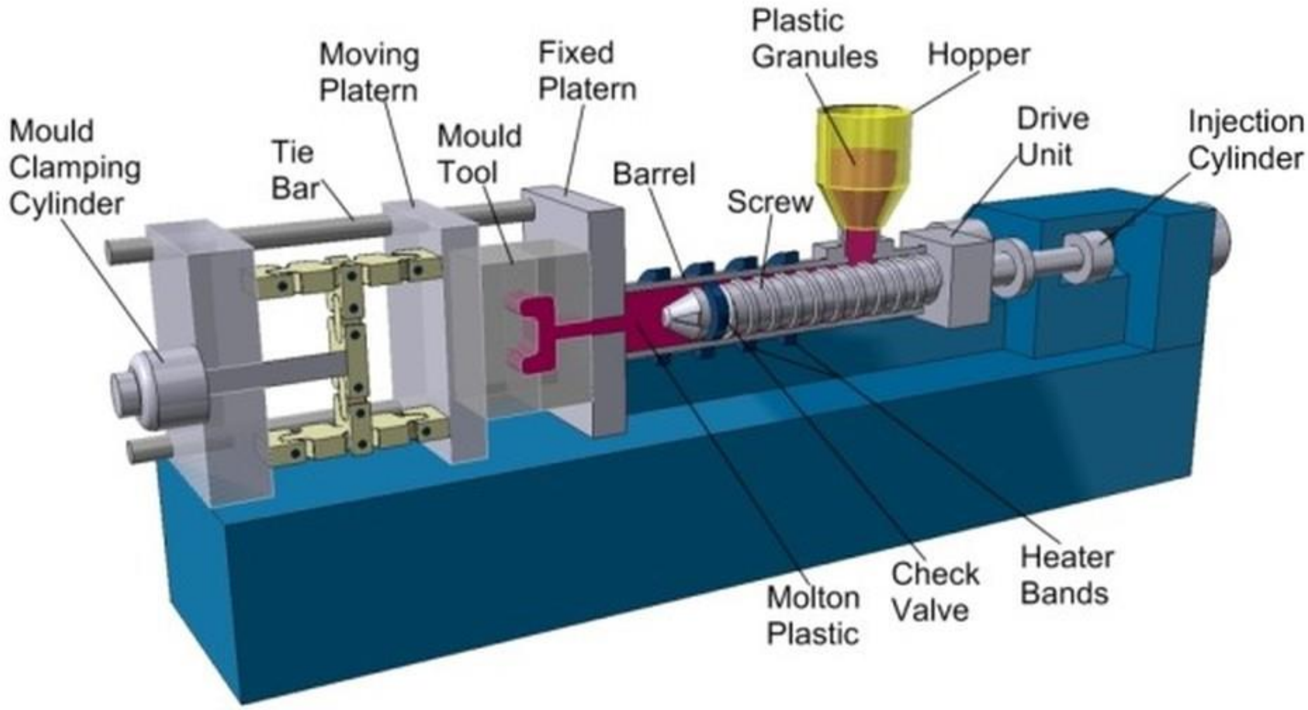

***Figure 1**. Injection Molding Process*

let's dive a bit deeper into the main components and characteristics of the injection molding process as depicted in the image:

1. **Plastic Granules:** These are the raw material for the injection molding process. The granules are typically made of thermoplastic or thermosetting plastic, chosen based on the desired properties of the final product.
2. **Hopper:** This is where the plastic granules are loaded. It's a funnel-shaped container that directs the granules down into the barrel. In some machines, the hopper is equipped with devices to dry the plastic before melting, as moisture can affect the quality of the molded product.
3. **Drive Unit:** This component is responsible for the movement of the screw and can consist of a hydraulic or electric motor. The drive unit provides the torque needed to rotate the screw, which in turn pushes the plastic through the heated barrel.

4. **Heater Bands:** These are positioned along the barrel and are responsible for heating the barrel externally to melt the plastic granules. They are controlled precisely to maintain the correct temperature for melting the specific type of plastic being used.
5. **Screw:** The screw is a crucial component inside the barrel that turns and moves the plastic granules forward. As it rotates, it also mixes and melts the granules into a homogeneous molten mass.
6. **Barrel:** This is the chamber where the plastic is melted. It houses the screw and is designed to withstand high pressures and temperatures.
7. **Injection Cylinder:** This mechanism pushes the screw forward, injecting the molten plastic into the mold cavity with the required force.
8. **Check Valve:** Also known as a non-return valve, it ensures that the molten plastic does not flow back into the barrel when the screw moves forward to inject the plastic into the mold.
9. **Mold Tool:** This consists of precisely engineered metal plates with cavities shaped as the final product. When the molten plastic is injected, it fills these cavities, taking on the desired form.
10. **Fixed Pattern:** This part of the machine holds one side of the mold in place.
11. **Moving Pattern:** Attached to the mold clamping cylinder, this plate can move to open and close the mold.
12. **Tie Bar:** These are the steel rods that guide the moving platen and help to align the mold accurately as it opens and closes.
13. **Mold Clamping Cylinder:** These hydraulic cylinders control the opening and closing of the mold. They apply a force sufficient to keep the mold closed against the pressure of the molten plastic being injected.

These components work in concert during the injection molding process to heat, convey, and inject molten plastic into a mold, where it cools and solidifies into the final part. The process is highly automated and can produce complex parts with high precision and repeatability.

Injection molding is celebrated for its ability to produce large volumes of parts with high precision, repeatability, and a high level of detail. Additionally, it allows for the manufacturing of complex shapes that would be difficult or economically unfeasible with other fabrication methods. Despite the initial high cost of mold design and creation, the efficiency and scalability of injection molding make it a cost-effective solution for mass production, benefiting from economies of scale.

Procedure

This study examines the injection molding simulation side of things. Because it allows engineers to see how well their mold and injection setup is working, simulation is a crucial phase in the process. The simulation should notify the engineer if there are any clear problems.

Designing of CAD model

SolidWorks, a leading 3D modeling software, was employed to develop a 3D model for this initiative. The design endeavor aimed to craft a unique shuriken that would serve as an exceptional addition to a collection of personal or martial arts accessories. The shuriken was meticulously engineered to ensure not only an aesthetically pleasing form but also functional durability and balance. Included are precise measurements and multiple images of the bespoke shuriken to provide a comprehensive understanding of the product.

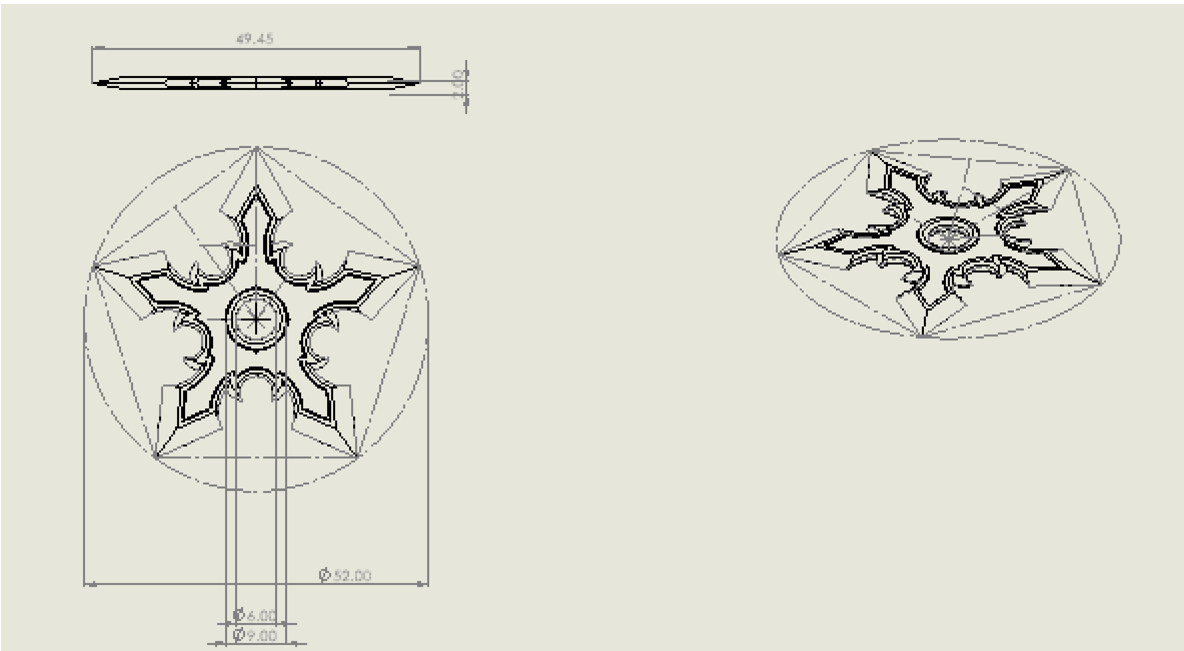

Figure 2. *Shuriken 2D drawing*

Simulation of Injection Molding Process

After the CAD model is developed, it is saved as a ".stl" file and imported into simulation software like Autodesk Moldflow Adviser to begin the simulation process. This stage is critical for identifying the optimal conditions for the injection molding process, including material selection, melt temperature, mold temperature, and injection location. The simulation helps in predicting the fill time, injection pressure, and quality of the molded part, allowing for adjustments to be made before the actual molding process begins.

Trial 01

Release version	2024
Study name	SHURIKEN_study
Study location	D:\DREXEL\Winter 2023 -2024\MEM 687 MP1 \finalproject\shuriken_study.sdy
Part name	SHURIKEN.STL
Model suitability	The imported model is thick and chunky, and is appropriate for 3D analysis.
Analysis resolution	Level 0 (Standard)
Material	
Material manufacturer	Generic Default
Material trade name	Generic PP
Environmental impact	
Melt temperature	220.0 (C)
Mold temperature	50.0 (C)
Injection locations	5
Max. machine injection pressure	180.000 (MPa)
Injection time selected	Automatic
Velocity/pressure switch-over	Automatic

Figure 3. *Initial parameters*

In the initial trial, the placement of the injection pins is distributed across five different points, as illustrated in the figure provided. The subsequent observations will focus on various parameters such as fill time and the overall quality of the mold. The starting conditions, encompassing the type of material used, as well as the melting and mold temperatures, are documented under the initial parameter settings.

Figure 4. *Injection pin locations*

Results:

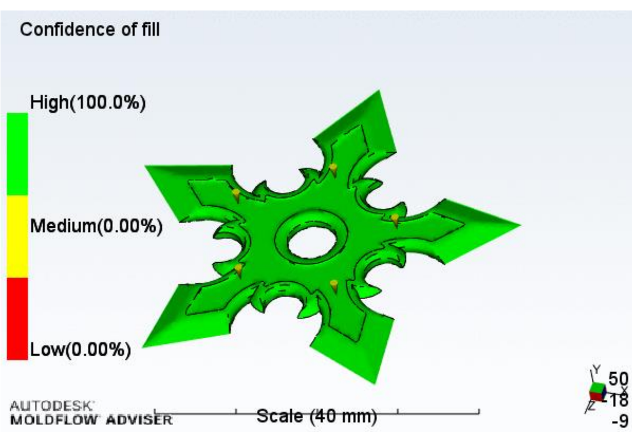

Figure 5. *Confidence of fill*

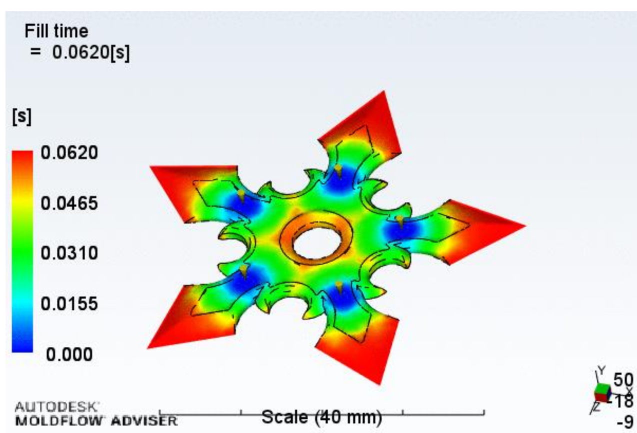

Figure 6. *Fill Time*

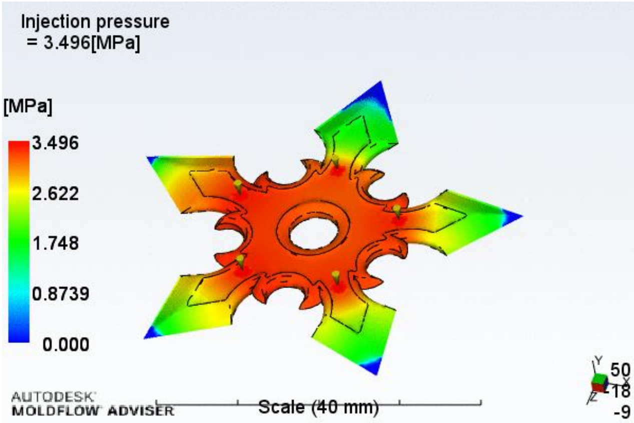

Figure 7. *Injection Pressure*

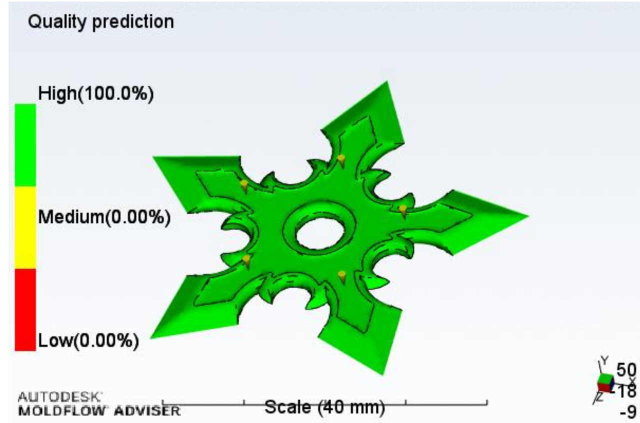

Figure 8. *Quality Prediction*

Your part can be filled easily with acceptable quality using the current injection locations.

Actual filling time	0.06 (s)
Actual injection pressure	3.496 (MPa)
Clamp force area	1.7655 (cm^2)
Max. clamp force during filling	0.049 (tonne)
Velocity/pressure switch-over at % volume	99.68 (%)
Velocity/pressure switch-over at time	0.06 (s)
Estimated cycle time	9.78 (s)
Total part weight	0.855 (g)
Shot volume	1.1230 (cm^3)

Clamp force estimate during packing using:	
20% of the injection pressure	0.013 (tonne)
80% of the injection pressure	0.050 (tonne)
120% of the injection pressure	0.075 (tonne)

Figure 9. *Trial 01 Results*

The results from an injection molding simulation state that the part can be filled easily and with acceptable quality at the current injection locations. Here are the key results from the simulation:

- The actual filling time is very quick at 0.06 seconds.
- The actual injection pressure required is moderate at 3.496 MPa.
- The clamp force area is noted to be 1.7655 cm².
- The maximum clamp force during filling is relatively low at 0.049 tonnes.
- The velocity/pressure switch-over occurs at 99.68% of the volume being filled, which is almost at the end of the filling phase.
- The switch-over time matches the filling time at 0.06 seconds.
- The estimated cycle time for the entire process is short, at 9.78 seconds.
- The total part weight is light, at 0.855 grams.
- The shot volume, representing the total volume of material injected, is 1.1230 cm³.

Additionally, the clamp force estimates during packing, based on varying percentages of the injection pressure, are given as:

- At 20% of the injection pressure, the estimated clamp force is 0.013 tonnes.
- At 80% of the injection pressure, the force is 0.050 tonnes.
- At 120% of the injection pressure, it increases to 0.075 tonnes.

These results suggest that the current injection molding parameters and locations are effective for producing the part with the equipment and conditions simulated.

Trial 02

In Trial 02, we will explore the effects of a material change on the injection molding process outcomes by switching to a Polycarbonate/Acrylonitrile Butadiene Styrene (PC/ABS) blend. This material amalgamates the robustness and thermal resistance of polycarbonate with the pliability and ease of processing inherent in ABS. The injection locations will remain unaltered to directly assess how the new material's properties contrast with the previous trial's results.

Release version	2024
Study name	SHURIKEN_study trial 02
Study location	D:\DREXEL\Winter 2023 -2024\MEM 687 MP1 \finalproject\shuriken_study_trial_02.sdy
Part name	SHURIKEN.STL
Model suitability	The imported model is thick and chunky, and is appropriate for 3D analysis.
Analysis resolution	Level 0 (Standard)
Material	
Material manufacturer	Generic Shrinkage Characterised Material
Material trade name	Generic PC+ABS
Environmental impact	
Melt temperature	255.0 (C)
Mold temperature	60.0 (C)
Injection locations	5
Max. machine injection pressure	180.000 (MPa)
Injection time selected	Automatic
Velocity/pressure switch-over	Automatic

Figure 10. *Initial Parameters for Trial 02*

Results:

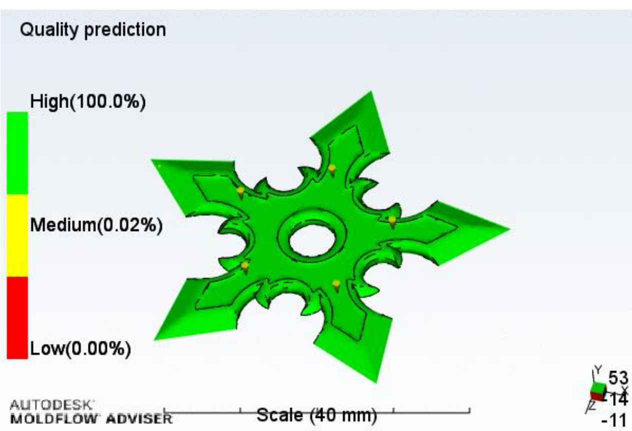

Figure 11. *Quality Prediction*

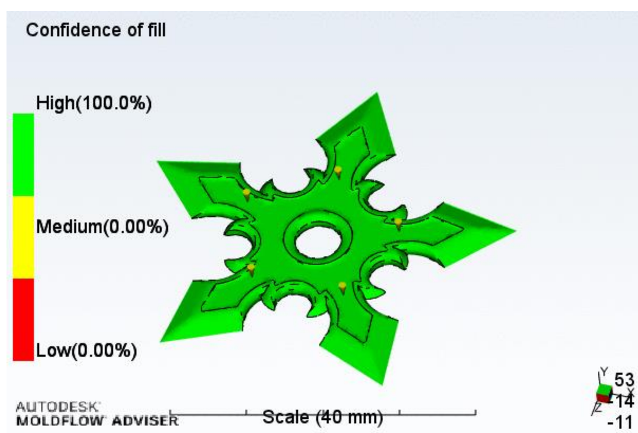

Figure 12. *Confidence of fill*

Your part can be filled easily with acceptable quality using the current injection locations.

Actual filling time	0.21 (s)
Actual injection pressure	8.763 (MPa)
Clamp force area	1.7655 (cm^2)
Max. clamp force during filling	0.143 (tonne)
Velocity/pressure switch-over at % volume	99.50 (%)
Velocity/pressure switch-over at time	0.20 (s)
Estimated cycle time	14.11 (s)
Total part weight	1.134 (g)
Shot volume	1.1230 (cm^3)

Clamp force estimate during packing using:	
20% of the injection pressure	0.032 (tonne)
80% of the injection pressure	0.126 (tonne)
120% of the injection pressure	0.189 (tonne)

Figure 13. *Trial 02 Result*

The results from Trial 02 of an injection molding process using a new material while keeping the injection locations unchanged. Here is an analysis of the provided data:

- The actual filling time is now longer at 0.21 seconds.
- The actual injection pressure has increased to 8.763 MPa.
- The clamp force area remains the same at 1.7655 cm².
- There is an increase in the maximum clamp force during filling to 0.143 tonnes.
- The velocity/pressure switch-over occurs at 99.50% of the volume, which is slightly less than the previous trial.
- The velocity/pressure switch-over time is slightly longer, now at 0.20 seconds.
- The estimated cycle time has increased to 14.11 seconds.
- The total part weight is now heavier at 1.134 grams.
- The shot volume remains consistent at 1.1230 cm³.

The clamp force estimates during packing, based on different percentages of the injection pressure, are also higher:

- At 20% of the injection pressure, the clamp force is 0.032 tonnes.
- At 80% of the injection pressure, the clamp force is 0.126 tonnes.
- At 120% of the injection pressure, it rises to 0.189 tonnes.

Comparing these results to those from the previous trial, it is evident that the new material requires a longer fill time and higher injection pressure, suggesting that it has a higher viscosity or requires more force to fill the mold. The cycle time is also longer, and the part ends up being heavier, indicating a possibly denser material. The increased clamp force during packing reflects the higher injection pressure needed for this material. Despite these changes, the message indicates that the part can still be filled easily with acceptable quality at the current injection locations.

Discussion

The simulation results from the injection molding trials provide insightful data on the behavior of two different materials under identical processing conditions. The first trial, using a standard material, exhibited very rapid filling times and lower injection pressures, indicating a material with a lower viscosity and ease of flow. The switch-over from injection to packing phase occurred almost at the end of the filling process, suggesting that the material fills the mold efficiently before additional pressure is applied to pack and cool the part. This resulted in a short cycle time, beneficial for mass production.

In contrast, the second trial with a PC/ABS blend showed a longer filling time and a substantially higher injection pressure, which suggests that this material has a higher viscosity and requires more force to fill the mold. The slight delay in the switch-over time and the increased cycle time imply that the material's different properties necessitate a more careful balancing of the injection molding parameters to achieve the desired part quality. The increased weight of the part might be indicative of the material's higher density or different flow characteristics that result in a slightly larger finished product.

Overall, these simulation outcomes are crucial for understanding the specific requirements of different materials and for adjusting the injection molding process parameters accordingly. The ability to fill the part easily and with good quality remains consistent, which is promising for the feasibility of using either material in a production environment. However, the changes in the process parameters highlight the importance of tailoring the injection molding process to the material's properties to optimize cycle time, material usage, and part quality.

Learning outcomes

From the injection molding project, some valuable learnings and insights can be summarized as follows:

1. Material Behavior: The properties of different materials, like Generic PP and PC/ABS blends, greatly influence the injection molding process. Parameters such as melt temperature, mold temperature, and injection pressure must be adjusted to accommodate the material's characteristics to ensure the quality of the final product.
2. Simulation Utility: The use of simulation software, such as Moldflow, is essential in predicting the outcomes of the injection molding process before actual production. This helps in identifying potential issues and saves cost and time by avoiding trial-and-error in physical trials.
3. Design Considerations: The design of the part, including the placement of injection locations, greatly affects the mold filling, cooling, and the overall quality of the product. Design adjustments might be necessary to optimize the manufacturing process.
4. Process Optimization: There is a critical balance between injection pressure, filling time, and cycle time that needs to be achieved for an efficient molding process. Optimizing these parameters can lead to improved quality and production efficiency.
5. Quality Assessment: Parameters such as fill time, part weight, and clamp force provide indicators of the potential quality of the molded part. Analyzing these can inform decisions to modify process parameters for better results.
6. Trial Comparisons: Conducting multiple trials with variations in material and process parameters allows for a comparative analysis that can lead to a deeper understanding of the influences on part quality and molding efficiency.
7. Mold Design and Mechanics: The importance of the mechanical design of the mold itself, such as venting, cooling channels, and gate location, is emphasized, as these factors are critical to the success of the molding process.
8. Regulatory Compliance: For certain products, understanding and adhering to relevant regulatory requirements, like Export Control regulations, can be critical to the project's execution and the product's marketability.
9. Economic Efficiency: Injection molding projects should not only focus on the technical aspects but also consider economic efficiency, including material costs, production rate, and energy consumption.

Filling process snapshots

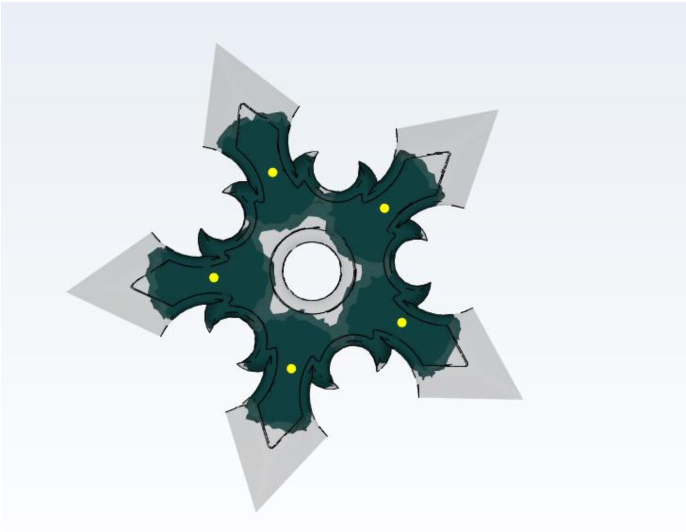

Figure 14. *Filled around 55%*

Figure 15. *Filled around 85%*

Figure 16. *Completely filed*