



Vietnamese-German University

## **COPYRIGHT WARNING**

This paper is protected by copyright. You are advised to print or download **ONE COPY** of this paper for your own private reference, study and research purposes. You are prohibited having acts infringing upon copyright as stipulated in Laws and Regulations of Intellectual Property, including, but not limited to, appropriating, impersonating, publishing, distributing, modifying, altering, mutilating, distorting, reproducing, duplicating, displaying, communicating, disseminating, making derivative work, commercializing and converting to other forms the paper and/or any part of the paper. The acts could be done in actual life and/or via communication networks and by digital means without permission of copyright holders.

The users shall acknowledge and strictly respect to the copyright. The recitation must be reasonable and properly. If the users do not agree to all of these terms, do not use this paper. The users shall be responsible for legal issues if they make any copyright infringements. Failure to comply with this warning may expose you to:

- Disciplinary action by the Vietnamese-German University.
- Legal action for copyright infringement.
- Heavy legal penalties and consequences shall be applied by the competent authorities.

The Vietnamese-German University and the authors reserve all their intellectual property rights.





RUHR-UNIVERSITÄT BOCHUM

**MechEng**  
Mechanical Engineering



Vietnamese-German University

# DESIGN, SIMULATION AND ANALYSIS FOR INJECTION MOLDING OF PLASTIC ICE TRAY

BACHELOR THESIS



Vietnamese-German University

BINH DUONG, NOVEMBER 2022

**Submitted by:** TRAN KIEN QUOC

**RUB Student ID:** 19220558

**VGU Student ID:** 14549

**Supervisor:** Prof. Dr. Nguyen Quoc Hung

**Co-supervisor:** Nguyen Phat Tai

# Design, Simulation and Analysis for Injection Molding of Plastic Ice Tray

A Thesis Presented

By

**Tran Kien Quoc**

Submitted to the department of Mechanical Engineering of the

***RUHR-UNIVERSITÄT BOCHUM and VIETNAMESE-GERMAN***



Of the requirement for the degree of

**BACHELOR IN MECHANICAL ENGINEERING**

Nov 2022

Major: Mechanical Engineering

Affirmation in lieu of oath

Tran, Kien Quoc

Matriculation Number: 14549

Title of Thesis: Design, Simulation and Analysis for Injection Molding of Plastic Ice Tray

I hereby declare in lieu of oath that I have produced the aforementioned thesis independently and without using any other means except the aids listed. Any thoughts directly or indirectly taken from somebody else's sources are made discernible as such.

To date, the thesis has not been submitted to any other board of examiners in the same or a similar format and has not been published yet.



Vietnamese-German University

Ho Chi Minh City, 7<sup>th</sup> November, 2022

A handwritten signature in black ink, appearing to read 'Kien Quoc', is written over a horizontal line. The signature is fluid and cursive.

Tran Kien Quoc

## ABSTRACT

Plastic items are becoming increasingly popular and diverse in the current era of industrialization and modernization in the country, progressively taking over the market. Almost every day, it is easy to come into contact with plastic products. And, as science and technology continue to advance, plastic products will evolve further, gradually upgrading production methods, types of machinery, and operating lines to suit consumer demands. Brass, pots, phone covers, motorcycles, pipes, medical equipment, and scientific research are all created from plastic. With the ongoing expansion of the plastics sector, the injection molding technology business is also evolving. As a result, it is vital to conduct research on plastic injection mold expertise in order to increase production and economic efficiency.

Understanding market needs and the development of plastic consumer products led us to choose the topic "Design, Simulation, and Analysis for Injection Molding of Plastic Ice Tray". The design and analysis were carried out for the parting surface, the runner system, the forming components, the ejector system, and the cooling system.



Vietnamese-German University

## ACKNOWLEDGMENT

This graduation thesis is a summary of the process of studying at school and preparing for real-world employment and production. I put a lot of effort into implementing and finishing the graduation thesis and was fortunate to receive a lot of attention and aid from the school and a wonderful teacher, with the help and encouragement of family and friends. These feelings give me confidence and excitement for my work, allowing me to learn a lot and successfully complete this project.

I would like to express my deepest gratitude to

- Prof. Doc. NGUYEN QUOC HUNG, the lead supervisor, who supported and encouraged me during the thesis's implementation.
- Senior NGUYEN THANH PHONG has provided all necessary documents to finish the graduation thesis.
- The lecturers at VGU's Faculty of Mechanical Engineering have given me the knowledge I need to complete the project.
- The director of Lap Phuc Precision Mold Co., who has offered me the opportunity to intern at the company.
- Finally, I'd like to thank my family for their tremendous emotional support.


THANK YOU VERY MUCH.

# TABLE OF CONTENTS

ABSTRACT.....	3
ACKNOWLEDGMENT .....	4
TABLE OF CONTENTS.....	5
LIST OF TABLES.....	8
LIST OF FIGURES .....	8
<b>1. INTRODUCTION .....</b>	<b>11</b>
<b>1.1. Overview .....</b>	<b>11</b>
<b>1.2. Objective .....</b>	<b>11</b>
<b>1.3. Research Methods.....</b>	<b>12</b>
<b>2. THEORETICAL BASIS.....</b>	<b>13</b>
<b>2.1. Overview of mold and injection molding technology .....</b>	<b>13</b>
<b>2.1.1. The concept of mold.....</b>	<b>13</b>
<b>2.1.2. Technical requirements for mold sets.....</b>	<b>13</b>
<b>2.1.3. Injection molding cycle.....</b>	<b>14</b>
<b>2.2. Basic components of a mold base. ....</b>	<b>15</b>
<b>2.2.1. Two-plate mold base .....</b>	<b>15</b>
<b>2.2.2. Three-plate mold base .....</b>	<b>17</b>
<b>2.3. General properties of plastic materials used in injection molding.....</b>	<b>18</b>
<b>2.3.1. Definition and Characteristics.....</b>	<b>18</b>
<b>2.3.2. Polypropylene (PP) .....</b>	<b>18</b>
<b>2.3.3. Polyethylene (PE) .....</b>	<b>19</b>
<b>2.3.4. Poly Acrylonitrile Butadiene Styrene (ABS) .....</b>	<b>20</b>
<b>2.3.5. Polyvinyl chloride (PVC).....</b>	<b>20</b>
<b>2.3.6. Polystyrene (PS) .....</b>	<b>21</b>
<b>2.4. Feeding System.....</b>	<b>22</b>
<b>2.4.1. Sprue .....</b>	<b>22</b>

2.4.2.	Runner System .....	23
2.4.3.	Gate .....	26
2.5.	Venting System.....	27
2.6.	Guide System.....	29
2.6.1.	Locating ring .....	29
2.6.2.	Guide pins and Guide bushing .....	30
2.7.	Cooling System.....	31
2.7.1.	Serial cooling channels .....	32
2.7.2.	Parallel cooling channels .....	33
2.8.	Ejector System.....	33
2.8.1.	Ejector pins.....	34
2.8.2.	Other components.....	34
3.	<b>DESIGN, SIMULATION AND ANALYSIS OF PLASTIC PRODUCT .....</b>	<b>36</b>
3.1.	Overview .....	36
3.2.	Product design requirements .....	36
3.3.	<b>Computer Aided Design and Computer Aided Engineering .....</b>	<b>36</b>
3.3.1.	NX 11.....	37
3.3.2.	Easy Fill Advance.....	37
3.3.3.	NX Mold Wizard.....	38
3.4.	Product design using NX 11 .....	38
3.5.	Material flow analysis using NX 11 Easy Fill Advance .....	43
3.5.1.	Overview .....	43
3.5.2.	Initialize project .....	43
3.5.3.	Best gate locations analysis .....	44
3.5.4.	Feeding system design .....	47
3.5.5.	Filling rate analysis .....	48
3.5.6.	Temperature analysis .....	49



3.5.7.	<b>Pressure analysis</b> .....	50
3.5.8.	<b>Clamping force analysis</b> .....	51
3.5.9.	<b>Defection analysis</b> .....	53
3.5.10.	<b>Analytical summary table</b> .....	56
<b>4.</b>	<b>STRUCTURAL DESIGN OF INJECTION MOLD</b> .....	<b>57</b>
4.1.	<b>Initialize project</b> .....	57
4.2.	<b>Parting Surface</b> .....	61
4.3.	<b>Splitting Core and Cavity</b> .....	63
4.4.	<b>Mold base selection</b> .....	63
4.5.	<b>Cooling system</b> .....	65
4.6.	<b>Ejector system</b> .....	68
4.7.	<b>Venting system</b> .....	70
4.8.	<b>Final product</b> .....	71
<b>5.</b>	<b>CONCLUSION</b> .....	<b>72</b>
	 Vietnamese-German University	
	<b>BIBLIOGRAPHY</b> .....	<b>73</b>

## LIST OF TABLES

Table 2.1: Type of gate and its application.....	27
Table 2.2: Air vent depth for each type of material.....	29
Table 2.3: Basic design cooling channel size (mm) .....	32
Table 3.1: Summary table.....	56

## LIST OF FIGURES

Figure 2.1: Injection molding cycle.....	14
Figure 2.2: Two-plate mold base .....	16
Figure 2.3: Three-plate mold base .....	17
Figure 2.4: Most common plastics processed by injection molding.....	22
Figure 2.5: Sprue .....	22
Figure 2.6: Types of runner shape .....	23
Figure 2.7: Cold runner system.....	24
Figure 2.8: Hot runner system.....	25
Figure 2.9: Vent system basic design .....	28
Figure 2.10: Locating ring in the injection molding system.....	30
Figure 2.11: Guide pin and guide bushing.....	31
Figure 2.12: Serial cooling channel .....	32
Figure 2.13: Parallel cooling system.....	33
Figure 2.14: Ejector system .....	34
Figure 2.15: Some ejector pins design.....	34
Figure 3.1: Create the block.....	39
Figure 3.2: Create the concave of the Ice Tray.....	39
Figure 3.3: Edge blending.....	40
Figure 3.4: Create pattern .....	40
Figure 3.5: Create the shell .....	41
Figure 3.6: Edge blending the rim .....	41
Figure 3.7: Create datum plane.....	41

Figure 3.8: Sketch the base of the ribs .....	42
Figure 3.9: Create the ribs.....	42
Figure 3.10: Edge blending the ribs .....	42
Figure 3.11: Final Ice Tray product .....	43
Figure 3.12: Select Cavity and Material Setting .....	44
Figure 3.13: Flow length/Thickness Ratio spectrum .....	45
Figure 3.14: One gate L/t ratio.....	45
Figure 3.15: Two gates L/t ratio .....	45
Figure 3.16: NX Easy Fill Advanced automatic gate location recommendation .....	46
Figure 3.17: Gate design.....	46
Figure 3.18: Recalculate L/t ratio of new gates location .....	47
Figure 3.19: Runner design.....	48
Figure 3.20: Sprue design .....	48
Figure 3.21: Melt Front Time analysis .....	49
Figure 3.22: Temperature analysis.....	50
Figure 3.23: Pressure analysis.....	50
Figure 3.24: Clamp Force analysis .....	52
Figure 3.25: Air trap analysis .....	54
Figure 3.26: Weld Line analysis .....	55
Figure 3.27: Volumetric Shrinkage analysis.....	55
Figure 4.1: Check Region and Draft Angle Limit .....	57
Figure 4.2: Check Wall Thickness.....	58
Figure 4.3: Initialize Project .....	59
Figure 4.4: Setting coordinate system of the mold .....	59
Figure 4.5: Create Workpiece .....	60
Figure 4.6: Setting Mold Layout .....	61
Figure 4.7: Identify the faces of the product to each region .....	62
Figure 4.8: Define Regions .....	62
Figure 4.9: Create Parting Surface.....	63

Figure 4.10: Cavity and Core splitting.....	63
Figure 4.11: Mold base selection .....	64
Figure 4.12: Mold base created.....	64
Figure 4.13: Sprue bushing design .....	65
Figure 4.14: Locating ring design.....	65
Figure 4.15: Pattern channel design.....	66
Figure 4.16: Extend Channel tab .....	66
Figure 4.17: Cooling Circuits setting.....	67
Figure 4.18: O-rings insert.....	67
Figure 4.19: Final design of cooling channels .....	68
Figure 4.20: Ejector pin design.....	68
Figure 4.21: Design layout of ejector pins.....	69
Figure 4.22: Creating the Ejector pins .....	69
Figure 4.23: Trimmed ejector pins .....	70
Figure 4.24: Venting channel parameters .....	71
Figure 4.25: Venting system layout.....	71
Figure 4.26: Final product .....	71

# 1. INTRODUCTION

## 1.1. Overview

As technology progresses, the applications of plastics become more diversified. Plastics have developed from lightweight, low-cost materials like those found in plastic bottles to high-performance ones. Some polymers, such as nylon 66 and polyphenylene sulfide, currently outperform metals in terms of performance and can be utilized to replace metals in high-stress applications.

Injection molding is one of the most regularly utilized production methods for the different plastic products used on a daily basis. The mold is the most important item to analyze when studying the injection molding process. It gives the product the necessary shape, so creating the component of the product related with the mold demands thought.

Mold and product design nowadays necessarily require the use of computer design software such as NX 11, Solid Edge, and AutoCAD for part designing and assembly software for the aggregate design of the mold assembly, making mold manufacturing precise and accurate for the injection molding manufacturing process of plastics and allowing for a wide range of tolerances with production variations.

With the ongoing expansion of the plastics sector, the injection molding technology business is also evolving. As a result, it is vital to conduct study on plastic injection mold expertise in order to increase productivity and economic efficiency.

## 1.2. Objective

Following a review of the literature on basic plastic material analysis and the principles of mold design theories for injection molding, the ICE TRAY product part design, simulation, and analysis is carried out while keeping its unique identity as a plastic product part design, and the following specific objectives are satisfied.:

- Reasonable design, saving materials but still ensuring function.
- The details of the mold set after processing must ensure the required features such as dimensional accuracy, durability, bearing capacity, etc.
- Precise position relative to mold plates and standard parts.
- Satisfactory dimensional tolerances (in the dimensional tolerance table).

- The plastic materials used are suitable and have good aesthetics.

### **1.3. Research Methods**

- Method of observation: actual observations from available models, videos, and actual molds have been studied to provide a framework for the topic.  
Simultaneously observe the actual machining process and assemble the mold set.
- Methods of reference materials: documents related to the project, sources from the book library, the internet, teachers, and friends.
- Prediction: always make assumptions about damage and possible difficulties during the design and manufacture process, from which experience can be drawn to perfect the mold.



Vietnamese-German University

## **2. THEORETICAL BASIS**

### **2.1. Overview of mold and injection molding technology (Kazme, 2016)**

#### **2.1.1. The concept of mold**

A mold is an assembly of many parts assembled together to form a complete block capable of forming a product. Between the two mold plates, there will be a space and a certain shape, and that is the part that helps form the product in the injection molding process. It consists of two main mold parts:

- The cavity part (fixed mold): is attached to the fixed plate of the plastic injection machine.
- The core part (movable mold): is attached to the movable plate of the plastic injection machine.

Furthermore, the surface where the mold's pieces meet is known as the parting surface. Mold halves are seldom exactly aligned when they close. This results in a visible parting line on the molded object at the point where the cavity inserts, core inserts, and plastic mold come together. Any major deflection of the cavity insert away from the core insert may result in a gap through which plastic will flow and form a thin film of plastic termed "flash" at this site, causing cosmetic issues. As a result, the parting line should run along the bottom edge or non-visible, non-functional edge of the part.

#### **2.1.2. Technical requirements for mold sets**

Plastic injection molding requires a lot of requirements and focuses mainly on the following requirements: accuracy of shape and size, hardness, and smoothness of the detailed surfaces in the mold.

- Accuracy in shape and size.

Shape-size accuracy is the most important part of the mold set. It directly affects the product formation of the injection molding process. When processing mold plates, mold plates must ensure absolute accuracy. Any deviation can lead to damage during product injection. Therefore, it is necessary to have a technological cycle and a reasonable cutting mode when machining.

- Hardness and smoothness.

The hardness of the parts in the mold also affects the durability of the mold. The fact that the mold plates are too thin also affects the pressing process. The injection mold works continuously with high frequency and high pressure during the working process, so it can lead to abrasion or warping of the mold plates, causing the mold plates to be damaged. Therefore, it is necessary to choose the right material and choose the appropriate thickness for each mold plate to improve the durability of the mold.

### 2.1.3. Injection molding cycle (Xie, Shen, & Jiang, 2011)

Injection molding is the method of molding a polymer by injecting it under high pressure into a mold. This process's individual steps are fairly brief. In most cases, the injection molding process takes between 2 seconds and 2 minutes. It is divided into four stages.:

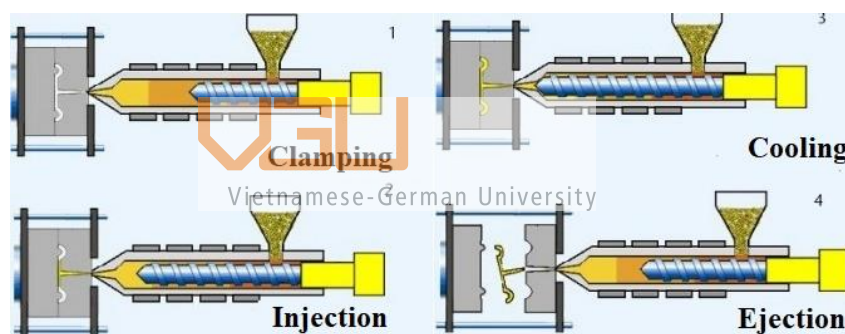


Figure 2.1: Injection molding cycle (Xie, Shen, & Jiang, 2011)

- Clamping phase

Before material is injected into the mold, a clamping unit must shut both halves of the mold. Then, with one half fixed and the other half adjustable, both sides are connected to an injection molding machine. The clamping unit then presses the pieces together, and as material is injected, both sections are tightly held together. Larger machines (with greater clamping force) require more time to seal and clamp the mold.

- Injection phase

The plastic pellets in the hopper are then fed into the injection unit's barrel. A screw inside the barrel moves the pellets forward. When plastic material enters the injection molding machine, it is melted after being squeezed by the screw, which generates a



significant amount of heat through friction. Melted plastic collected in front of the cylinder is heated by heater bands wrapped around the barrel to warm up the plastic pellets and maintain injection temperature. The volume of material injected is referred to as the "shot." When 95% to 99% of the mold has been filled, the injection time is complete.

- Cooling phase

As the plastic solidifies and cools, it will take the desired shape. The part may shrink slightly during cooling. After the cooling process is complete, the mold can be opened. The cooling time can be determined using the part's wall thickness and the plastic's thermodynamic parameters.

- Ejector phase

Ejection is the final stage. This is performed by employing an ejection system. When the mold opens, the part is pushed out. Force is required because the item shrinks and clings to the mold. The mold can be closed again after ejection and another shot injected to continue the process.



Vietnamese-German University

## **2.2. Basic components of a mold base.** (Menges, Michaeli, & Mohren, 2001)

### **2.2.1. Two-plate mold base**

It is a mold with only one parting surface. The two-plate mold is the most common type of mold. Compared with the three-plate mold, the two-plate mold is simpler, cheaper, and the injection molding cycle is shorter.

Advantages:

- The most simple and reliable design, as it normally has very little transfer details.
- Simple in making and working in production.
- Low price.

Disadvantage:

- The plastic entry point cannot be adjusted.
- The product requires low precision.

- Hard to automate.
- The plastic filling in the two-plate mold is uncontrolled.

For injection molding products that do not have high requirements for accuracy, choosing a two-plate mold with a simple design and a low price is the best choice.

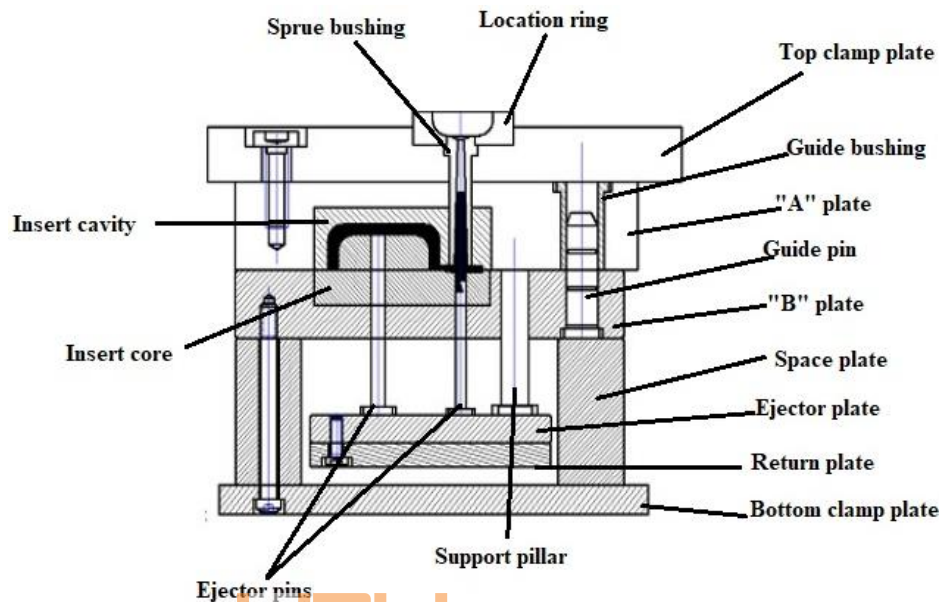


Figure 2.2: Two-plate mold base

Vietnamese-German University

Mold components function:

- Locating ring: center between the nozzle bush and the nozzle.
- Sprue bushing: guides the plastic from the injection molding machine into the plastic channels.
- “A” and “B” plates (cavity/fixed and core/moveable plate): holding the insert cavity and core.
- Insert cavity and core: shape the product.
- Top clamp plate: secure the fixed part of the mold to the press.
- Cooling plug: cooling channel’s inlet and outlet.
- Guide bushing: ensure the relative position between the “A” and “B” plate
- Guide pin: guide and position when the mold is closed.
- Ejector pin: push the product out of the mold.
- Return pin: return the ejector system to its original position when the mold closes
- Space plates: creating space for the ejector plate to work.

- Ejector plate: push the ejector pins.
- Return plate: holds the ejector pins.
- Support pillar: increase the durability of the mold during injection molding.

Sometimes, there is a cushion plate underneath the “B” plate in order to increase the strength of the injector housing.

### 2.2.2. Three-plate mold base

The injection mold with three plates has a cool runner mechanism. The runner is set on two plates, the cavity plate and the support plate, and when the mold is opened, it has an opening for the product to be taken out and another for the runner. As a result, if the product and channel are removed from the mold using a push system, two systems must be configured, making the mold structure more intricate and larger than the two-plate mold.

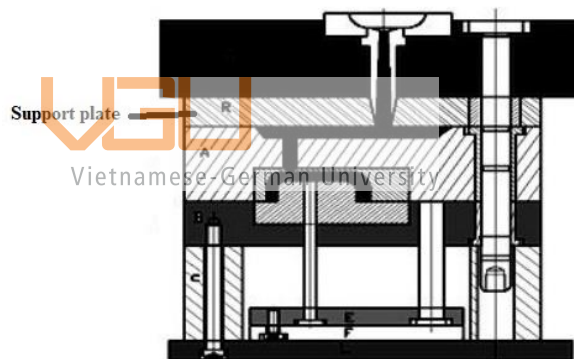


Figure 2.3: Three-plate mold base

Advantage:

- When the product and plastic channel are taken from the mold, they are always automatically separated.
- Can be used for large products requiring multiple nozzles or multi-cavity molds requiring multiple nozzles.

Disadvantage:

- The distance between the machine's nozzle and the mold cavity is significant.
- High price.
- Hard to design and manufacture.

## **2.3. General properties of plastic materials used in injection molding**

### **2.3.1. Definition and Characteristics**

Plastics are a type of synthetic or semi-synthetic material composed primarily of polymers. Because of their fluidity, plastics may be extruded, pressed, or molded into solid objects of various shapes. This versatility, combined with other features like as flexibility, cheap production costs, durability, and light weight, has resulted in widespread application.

Plastics are made from organic materials such as fossil fuels, petroleum, coal, natural gas, and plant materials. They are created in a variety of compositions based on their use and are usually classed as thermoplastics and thermosetting plastics. They are distinguished by their capability to be reshaped and converted into the required product through the use of various industrial plastic manufacturing methods such as molding, extrusion, piping, and pulling to extrusion by applying heat and pressure. These plastics are produced industrially through the polymerization process and are mostly constituted of synthetic resins, which are the polymers that comprise their structure as chained monomers. (Harper, 2000)



Vietnamese-German University

Each product or injection molding cycle has unique qualities and technical needs, such as plasticity, surface gloss, color, hardness, and so on. As a result, it is critical to select the proper type of plastic material to avoid flaws in the pressing process while still ensuring the product's mechanical and aesthetic requirements.

Some of the most popular injection molding materials being used in the market are as follows:

### **2.3.2. Polypropylene (PP)**

Because it does not allow chemicals to combine with food products, this thermoplastic injection molding material is widely utilized in the food storage and packaging business. Polypropylene (PP) is resistant to chemicals and moisture and may be washed in hot water without degrading. PP can possess exceptional impact strength, flexibility, and toughness. It should also be noted that PP is easily recycled, and because of its flexibility, it may be utilized to manufacture living hinges that can be bent repeatedly without ripping. (Williams, 2018)

Properties:

- High mechanical strength (tear and tensile strength), quite rigid, not as flexible as PE, not stretched. Especially the ability to tear easily when there is a small cut or puncture.
- Transparent, high surface gloss for high printability, clear print.
- Colorless, odorless, tasteless, non-toxic.
- Withstands temperatures higher than 1000 degrees Celsius.
- Has the property of repelling O<sub>2</sub>, steam, and other gases.

Application:

- Used as a single-layer packaging for food preservation
- Production of solid plastic containers
- Creating textile fibers for food and grain packaging

### **2.3.3. Polyethylene (PE)**

Polyethylene (PE) is a light thermoplastic molding material with excellent resistance, flexibility, and electrical insulation. It's not particularly strong or hard, but it's cheap. It is present in a variety of consumer plastic products, including milk bottles, medication and detergent bottles, plastic bags, and garbage cans. PE is also the most commonly used injection molding plastic for the manufacture of toys since it is non-toxic and can withstand a battering without complaint. (Williams, 2018)

Properties:

- Transparent, slightly translucent, with a glossy, flexible surface.
- Good resistance to water and steam.
- Poor resistance to O<sub>2</sub>, CO<sub>2</sub>, N<sub>2</sub> gas and grease.
- Withstands high temperatures (below 230 degrees Celsius) for a short time.
- Being inflated and damaged when in contact with aromatic oils or detergents such as Alcohol, Acetone, H<sub>2</sub>O<sub>2</sub> ...
- Absorbent holds the odor in the packaging itself, and this same odor can be absorbed by the food it is contained.

Application:

- Products that require mechanical tensile strength: Plastic hammers, electrical and thermal insulation materials, bathtubs, and water pipes.
- Products that need to be resistant to solvents and oils: solvent containers, bottles, packaging films.
- Products used for electrical insulation: making high-frequency resistant materials, insulating tapes.

#### **2.3.4. Poly Acrylonitrile Butadiene Styrene (ABS)**

ABS plastic is an engineering grade plastic and an opaque thermoplastic polymer. There are numerous benefits to using ABS. It's strong, has good dimensional stability, can withstand impacts and scratches, and is difficult to break. Furthermore, the low melting point makes it easy to mold. It's often used to make electronic parts like plastic wall socket guards, keyboard keys, and phone adaptors. Because ABS is a good insulator, it will not conduct electricity or emit gases if exposed to fire. These are critical issues for electrical device product developers. (Williams, 2018)

Properties:

- Opaque white, semi-transparent, has higher viscosity and impact strength than PS.
- Heat deformation temperature: 60-120 degrees Celsius. Flammable.

Application:

- In insulating products, in electronics and communications engineering (cases and internal components).
- In cryogenic engineering: As inner enclosures, inner doors and outer sheaths that are impacts resistant at cold temperatures.
- Injection molding products such as covers, keyboards, used in office machines, cameras...

#### **2.3.5. Polyvinyl chloride (PVC)**

PVC is a heat-sensitive polymer with poor injection molding characteristics. This is because PVC decomposition is easily induced by either an exceptionally high melt temperature or an excessively long heating period. (Williams, 2018)

Properties:

- Low thermal stability, flexible with the addition of plasticizers.
- Good weather resistance.
- Good dimensional stability.
- High durability, high aging resistance.
- Poor impact strength.

Application:

- Water pipes, thin films, plastic sheets.

### 2.3.6. Polystyrene (PS)

As with so many other things, there is a tradeoff to be aware of with polystyrene. On the one hand, PS is durable and hard. In the outdoors, it can withstand a lot of damage. However, this also means that it is not very environmentally friendly due to its difficulty in recycling. (Williams, 2018)

Properties:



- Amorphous.
- High strength, low impact resistance.
- Mechanical properties: colorless, transparent, easy to color, good elongation
- Low deformation temperature, produces black gas when burning.

Application:

- Cheap products, recycled plastic products such as cups and boxes.
- High frequency insulators are used to make electrical boxes, pipes, and insulating materials.

SPECIFICATIONS OF THE MOST COMMON PLASTICS PROCESSED BY INJECTION MOLDING							
MATERIAL NAME	SYMPOLE	DENSITY g / cm <sup>3</sup>	Glass-Fiber content % %	Average specific Heat Kj / (Kg*K)	Processing Temperature C-degree	Mold Temperature C-degree	Shrinkage %
1-PolyStyrene	PS	1.05		1.3	180-280	10-40	0.3-0.6
1-PS-High and Medium Impact	HI-PS	1.05		1.21	170-260	5-75	0.5-0.6
1-Styrene-Acrylo-Nitrile (b)	SAN	1.08		1.3	180-270	50-80	0.5-0.7
1-Acrylonitrile-Butadiene-Styrene	ABS	1.06		1.4	210-275	50-90	0.4-0.7
1-Acrylonitrile-Styrene-Acrylate (b)	ASA	1.07		1.3	230-260	40-90	0.4-0.6
2-Low density PolyEthylene	LDPE	0.954		2.0-2.1	160-260	50-70	1.5-5.0
2-High density PolyEthylene	HDPE	0.92		2.3-2.5	260-300	30-70	1.5-3.0
2-PolyPropylene	PP	0.915		0.84-2.5	250-270	50-75	1.0-2.5
2-PolyPropylene-GR	PPGR	1.15	30	1.1-1.35	260-280	50-80	0.5-1.2
2-PolyisoButylene	IB				150-200		
2-PolyMethylPentene	PMP	0.83			280-310	70	1.5-3.0
3-PolyVinylChloride, soft	PVC-soft	1.38		0.85	170-200	15-50	> 0.5
3-PolyVinylChloride, rigid	PVC-rigid	1.38		0.83-0.92	180-210	30-50	-0.5
3-PolyVinylDeneFluoride	PVDF	1.2			250-270	90-100	3-6
3-PolyTetraFluoroEthylene	PTFE	2.12-2.17		0.12	320-360	200-230	3.5-6.0
3-Fluorinated Ethylene Propylene	FEP						
3-Copolymer(b)							
4-PolyMethyl Methacrylate(Acrylic)	PMMA	1.18		1.46	210-240	50-70	0.1-0.8
5-PolyOxyMethylene (Acetal)	POM	1.42		1.47-1.5	200-210	> 90	1.9-2.3
5-PolyPhenylene Oxide	PPO	1.06		1.45	250-300	80-100	0.5-0.7
5-PolyPhenylene Oxide-GR	PPO-GR	1.27	30	1.3	280-300	80-100	< 0.7
6-Cellulose Acetate	CA	1.27-1.3		1.3-1.7	180-230	50-80	0.5

Figure 2.4: Most common plastics processed by injection molding (Wang, Chang, & Hsu, 2018)

## 2.4. Feeding System

### 2.4.1. Sprue

The sprue is the connection between the machine's nozzle and the runner channels, with the function of delivering the plastic flow from the machine's nozzle to the channel or right next to the cavity (for molds without runner). Sprue bushings are frequently utilized in the most regularly used sprue system for simple replacement and processing.

Choose the sprue bushing type based on:

- Product weight and chosen material.
- Mold base dimensions.
- Nozzle diameter.

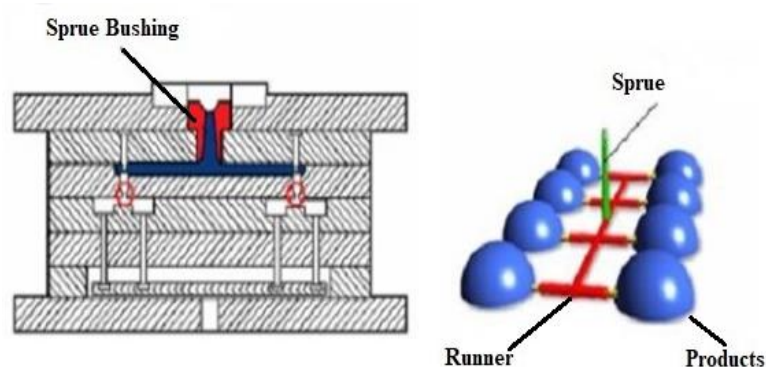


Figure 2.5: Sprue



## 2.4.2. Runner System

### 2.4.2.1. Overview

A runner channel is the connection between sprue and gate, with the task of putting molten plastic material into the mold cavity.

Runners can be formed in numerous ways. Some examples are full round, parabolic, trapezoid, wide trapezoid, half round, and quarter round. Full round runners are the most efficient. It has the least pressure drop per material unit volume. However, its biggest downside is that the runner must be cut into both A and B runner plates, which means that the two halves must fit together when closing the mold.

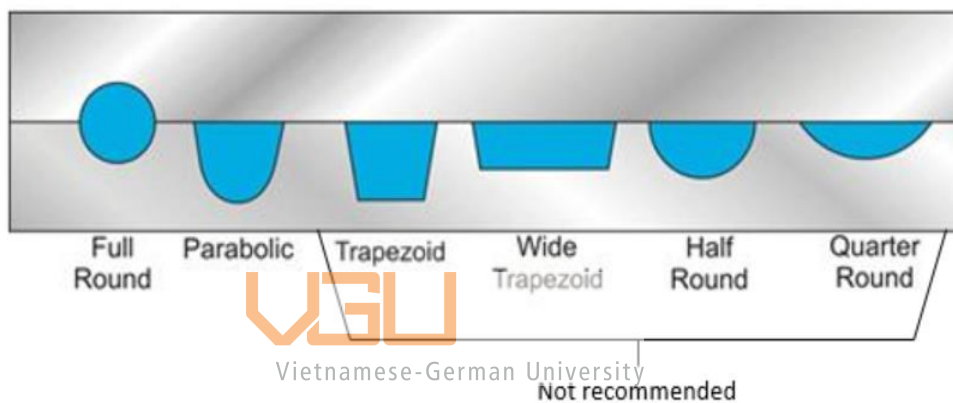


Figure 2.6: Types of runner shape (Amon, 2008)

There are a few rules that need to be followed when designing the runner:

- Minimize the change of runner cross-section.
- The plastic in the lead runner must exit the mold easily.
- The entire channel length should be as short as possible, avoiding pressure loss and heat loss during filling.
- Small enough to reduce scrap, shorten cooling time, and reduce clamping force. But it must be large enough to transfer a substantial amount of material to fill the cavity quickly and with little loss of pressure.

The runner channel can have a variety of cross-sections. However, selecting the best cross-section depends on a variety of circumstances. To choose the proper option during the design process, it is vital to pay attention to the specifications set forth, as well as the advantages and disadvantages of each type of section. Furthermore, to maintain

product stability, the length and diameter of the runner from the sprue to each cavity should be the same. It must be calculated and set so that the flow and pressure in the mold cavity are balanced. In addition, the round runner size can be calculated according to the following formulas:

$$D = \frac{W^{1/2} \cdot L^{1/4}}{3.7} \quad (2.1)$$

In there: D: Diameter of the runner (mm)

W: Weight of the product (g)

L: Length of the runner (mm)

In terms of systems, there are two types of primary runner systems: cold runner and hot runner.

#### 2.4.2.2. Cold runner system

Cold runner molds are typically made up of two to three plates. A nozzle injects molten thermoplastic into the mold, filling the runners that lead to the mold cavities. While the final result varies depending on whether a two-plate or three-plate mold was used, cold runner molds frequently produce a final item that contains the sprue, if not the sprue and the runner system. To reduce waste, the runner in both systems is reground and recycled on a regular basis.

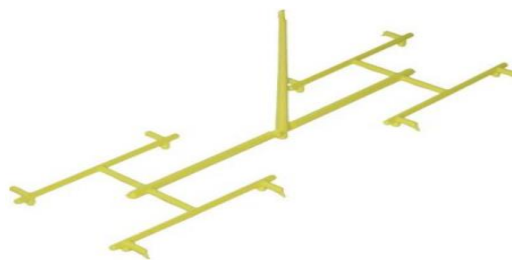


Figure 2.7: Cold runner system

Advantage:

- Low cost
- Easy to design, manufacture and maintain.

Disadvantage:

- Extreme high-pressure consumption.
- The smallest temperature variation results in a substantial viscosity difference and, as a result, a non-uniform plastic melt.
- Filling all of the cavities at once and in the same state is hard.
- Long cycle time and inconsistent fill time.

### 2.4.2.3. Hot runner system

The hot runners are a complicated mechanism that maintains a high enough temperature to allow the melt to flow while the mold cools and solidifies. A manifold system is utilized in hot runner molds to heat a mold made up of two plates. The hot runner system, as seen in Figure 2.8, consists of a heat manifold and nozzles along the runner, with an application for the system to distribute the plastic melt entering the mold to numerous nozzles that accurately meter to the injection locations in the cavities.

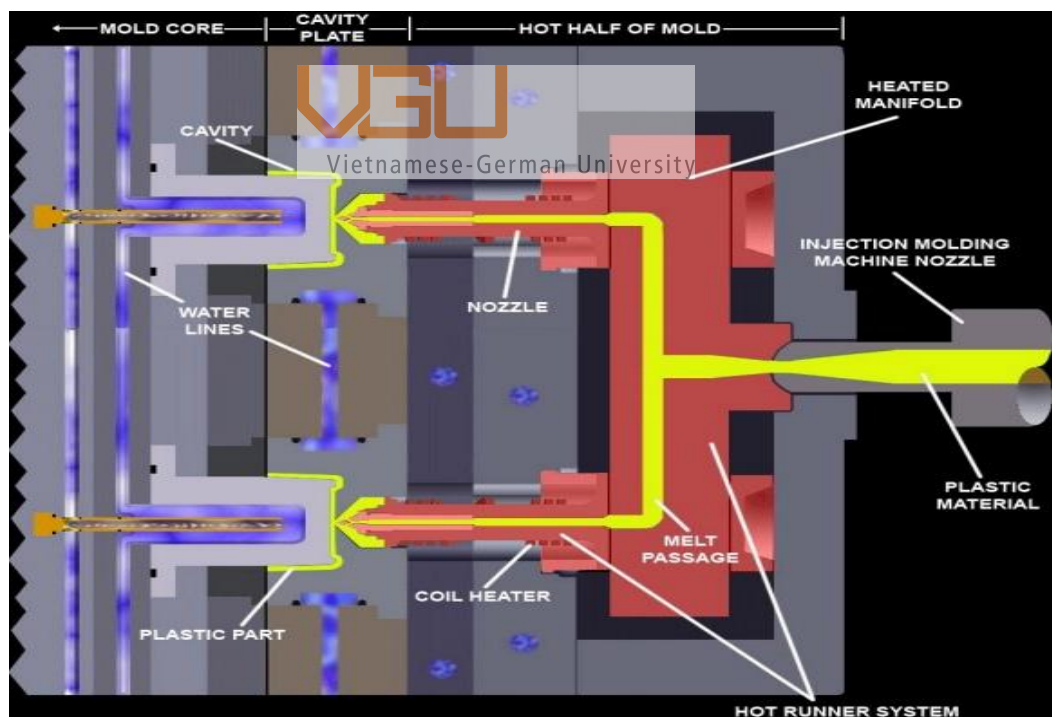


Figure 2.8: Hot runner system (Leecher, 2020)

Advantage:

- Uniform wall thickness.
- Because there is no melt loss, the energy and work input are reduced.

- Operation is simpler and entirely automatic.
- This results in a high-quality product.
- Weld lines are reduced or eliminated.
- High cavitation and large molding applications are possible.
- Micro molding is possible.

Disadvantage:

- Very expensive.
- Thermal isolation from the hot runner manifold is problematic.


### 2.4.3. Gate

The gate is a molten material flow path between the runner channels and the mold cavity. Plastic part quality is directly connected to gate design and gate position choices.

When designing a gate, it is necessary to pay attention to the following points:

- The gate needs to be positioned so that the plastic flows in
- where there is maximum to minimum wall thickness for the material to fill the product.
- The optimal gate position will create a smooth flow of plastic.
- Place the gate in an unimportant position of the product to increase the aesthetics of the product.
- The gate should be located in a position that can expel all the air from the vent without creating air bubbles in the product.
- Set the gate so that no weld lines are left, especially when using multiple gates.

Commonly used type of gate:

	Type of gate	Description
	Sprue gate	<p>The sprue gate is virtually like the lack of a gate: the sprue transports material directly into the mold chamber. Direct gates are simple to install, but they generate a lot of tensile tension surrounding the gate.</p>






	<p>Pin gate</p>	<p>This type is commonly used with three-plate molds or large cavity molds that need multiple nozzles or for molds with multiple cavities. The advantage of this type is that it is possible to arrange many nozzles into the mold cavity for large mold cavities, making the filling faster and better.</p>
	<p>Edge gate</p>	<p>The most prevalent gate type in injection molding is the edge gate. It is applied along the part's edge, leaving a noticeable line at the parting line.</p>
	<p>Fan gate</p>	<p>The fan gate is just a variable-width edge gate. This type of gate creates a smooth flow and allows the cavity to be filled quickly, making it very suitable for large and thick products. However, it is very difficult to cut the gate.</p>
	<p>Tunnel gate</p>	<p>The tunnel type nozzle is usually used for two-plate molds with multiple cavities. This type is also very common and has the advantage that it cuts itself when the product is pushed out of the mold. This type of gate can be put on the pattern lines or ribs to hide the traces of the gate.</p>
	<p>Cashew gate</p>	<p>A cashew gate is installed below the part line and is thus suited for automatic trimming. It has a one-of-a-kind design that arcs around the part. This virtually eliminates any touch with the product, making this a perfect choice for a smooth finish.</p>

Table 2.1: Type of gate and its application (Rosato, 2000)

## 2.5. Venting System

As the cavity is filled, the melts push the air in front of them to the boundaries of the cavity. As the temperature and air pressure rise, the surrounding polymer melts may burn or there may be resistance, preventing the trapped air in the end walls from entirely escaping. When the melt, on the other hand, splits into two gated system cavities or flows in a certain way, it generates an air gap where it meets the melt, which may result

in a hole in the part. The process of releasing air from a mold cavity is known as venting.

The first choice is the air vent located on the parting surface because it is easy to process and clean. These vents act as a bridge between the mold cavity and the outside environment to help bring air out of the mold cavity.

Air vents should be installed at various points throughout the runner system or at the molded part's boundary. Usually, those are the areas farthest from the gate, because when filling, the air is compressed and forms air traps, so it is arranged here. They are especially needed in the last filled positions. The number of air vents depends on the circumference of the part. The total circumference of the air outlet must be close to 30% of the parting line of the part to ensure the ability to escape.

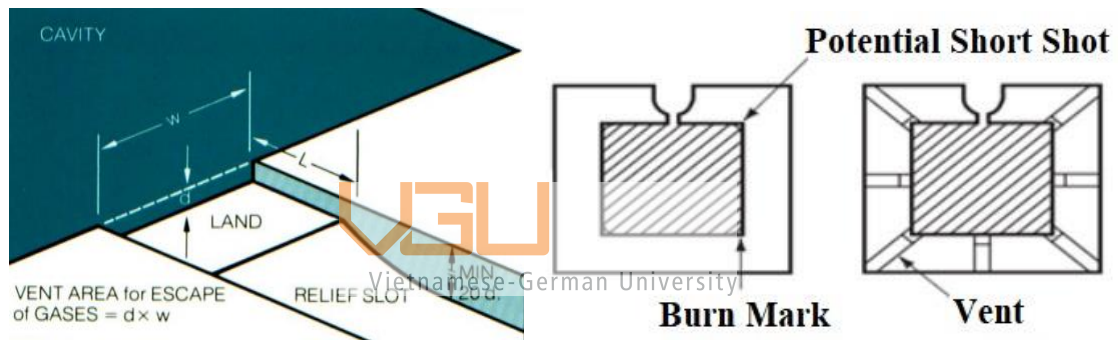


Figure 2.9: Vent system basic design (Designing Guides: Engineering Plastics., 2005)

Parameters needed to design the exhaust channel:

- Depth of vents:  $d$
- Width of the vent:  $W$
- Vent length:  $L$
- Depth of drain  $D = 20d$

The depth at the beginning of the vent area is usually small to prevent the material from flowing out. And depending on the viscosity of each type of plastic, there are different depths, given in the following table:

Material	Air vent depth (mm)	Material	Air vent depth (mm)
ABS	0.0254 ÷ 0.381	POLYPROPYLEN	0.0127 ÷ 0.0508
ACETAL	0.0127 ÷ 0.0254	PVC (HARD)	0.01524 ÷ 0.0254
ACRYLIC	0.0381 ÷ 0.0508	PVC (SOFT)	0.0127 ÷ 0.01778
CELLULOSE ACET, CAB	0.0254 ÷ 0.0381	POLYETHYLENE	0.0127 ÷ 0.03048
ETHYLENE VINYL ACET	0.0254 ÷ 0.0381	PET, PBT, POLYESTERS	0.0127 ÷ 0.01778
NYLON	0.00762 ÷ 0.0127	POLYSULFONE	0.0254 ÷ 0.0508

Table 2.2: Air vent depth for each type of material (Harper, 2000)

## 2.6. Guide System

### 2.6.1. Locating ring



Vietnamese-German University

High precision contact between the sprue bushing and the nozzle coupling are required. The nozzle usually cannot be sealed by sprue bushing, which causes grooves or gaps to appear in the contact between the nozzle and the sprue bushing, which interrupts the process and leaks material. The locating ring makes the connection between the sprue bushing and the nozzle concentric and tight.

Some technical requirements:

- The positioning ring is pre-fabricated according to the plastic injection mold manufacturer's standards.
- The positioning ring has a taper on the inside to allow the plastic nozzle on the press to attach to form a tight joint.
- Select the locating ring suitable for sprue bushing and nozzle, and also suitable for injection shaft.

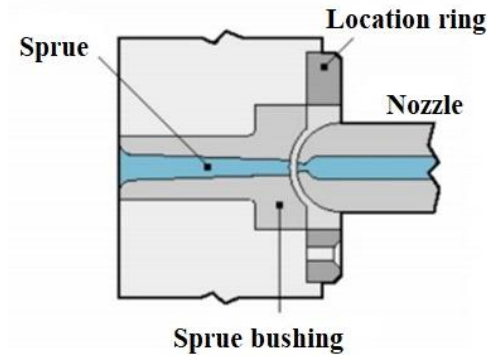


Figure 2.10: Locating ring in the injection molding system

### 2.6.2. Guide pins and Guide bushing

The guide pin is attached to the movable die plate and slides precisely into the guide bearing mounted on the fixed die plate.

The guide bushing has the effect of improving accuracy and reducing wear errors. In addition, the guide bushing reduces wear and is easy to replace and use.

The guide pin and the guide bushing have the function of ensuring firmly the assembly of the position between the two fixed and moving molds during operation.

Vietnamese-German University

Technical requirements:

- For the guide pin, the selected material must be able to achieve high external hardness to prevent abrasion and must have internal plasticity to prevent breakage during work.
- The length of the guide pin depends on the depth of the cavity
- The tip of the guide pin should be smaller and taper to make the initial assembly easy.
- For guide bushing, the common manufacturing materials are copper and brass. For large mass production, tempered steel is used.
- When manufacturing bushings, the most important technical requirements are the concentricity between the outer face and the hole face, as well as the perpendicularity between the outer face and the centerline.

Arrangement:



- If there is no obstruction to the product, the guide pin should be placed on the movable mold plate.
- Aim for the most reasonable layout, located in the four corners of the mold and corresponding to the cooling system of the mold.

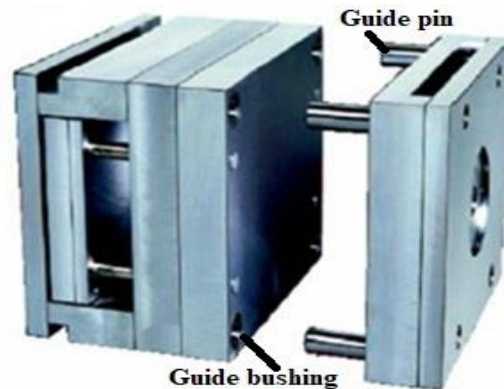


Figure 2.11: Guide pin and guide bushing

## 2.7. Cooling System

Cooling time accounts for about 60% of the mold cycle time, so how to reduce the cooling time but still ensure product quality is important. The melting temperature of the plastic put into the mold is usually about 150 to 300 degrees Celsius. When the plastic material is put into the mold at this high temperature, a large amount of heat from the plastic material is transferred to the mold and through the mold cooling system. If the cooling system, for some reason, has not been able to effectively deliver heat to the mold, the temperature in the mold will constantly increase, increasing the production cycle.

Purpose:

- Keep the mold at a stable temperature so that the plastic material can cool down evenly.
- Rapid cooling, avoiding the case that the heat does not cool in time, causing product deformation causing waste products.
- Reduce cycle time, increase production capacity.

Notes when designing cooling channels:

- The cooling channel should be close to the mold cavity for better heat dissipation, but the mechanical strength of the mold material should be considered.
- To avoid line breakage and poor heat exchange, the diameter of the water line should remain constant throughout the channel.
- It is recommended that the cooling channel be divided into several cooling loops. The length of the cold channel should not be too long, as this will cause pressure loss and temperature rise, increasing the temperature difference between the inlet and outlet by more than 3 degrees Celsius.
- The mold material's thermal conductivity is also important.

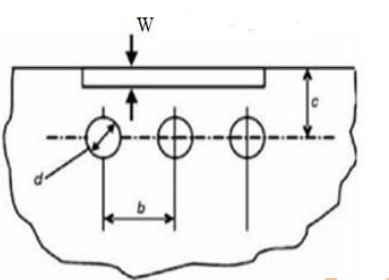
	W	d	c	b
	Part thickness	Cooling channel diameter	Distance between part and cooling channel	Distance between two cooling channels
	1-2	8-10	2-2.5d	2-3d
	2-4	10-12		
	4-6	12-14		

Table 2.3: Basic design cooling channel size (mm)

### 2.7.1. Serial cooling channels

In this design, which is the most common cooling channel construction, the coolant flows from the in and output manifolds in a single circuit connection. This type of system is commonly utilized in mold cooling systems because it maintains unchanged cross-sectional diameter and channel distance while allowing uniform mold cooling and turbulent flow for effective.

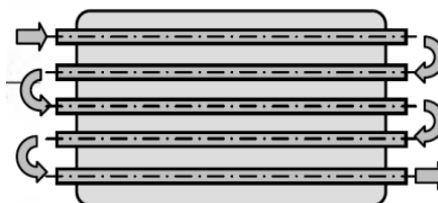


Figure 2.12: Serial cooling channel (Park & Dang, 2012)

Characteristics:

- Coolant flows through a connected circuit.
- There is a temperature difference between the inlet and outlet ports.
- There is a high pressure drop between the inlet and outlet ports.
- Any blockage in this circuit is easily identifiable.

### 2.7.2. Parallel cooling channels

Coolant flows in two directions from the supply manifold to the collecting manifolds through parallel tubes. The figure below illustrates the layout. Because of the arrangement, this type of cooling channel layout has a variable flow rate, causing the system's cooling capabilities to be unequal, resulting in uneven cooling of the mold.

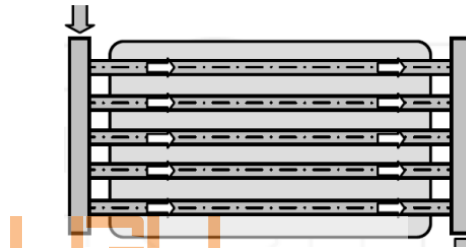


Figure 2.13: Parallel cooling system (Park & Dang, 2012)  
Vietnamese-German University

Characteristics:

- All circuits are provided with a common cooling source at the same temperature.
- There is a lower temperature difference through the mold.
- There is a lower temperature drop between inlet and outlet
- More uniform mold temperature.
- Blockages are harder to detect.

### 2.8.Ejector System

When the filling and cooling processes are finished, the ejector system is in charge of forcing the product out of the mold cavity. Because the product becomes trapped on the mold wall after cooling, a method to force the product out of the mold must be designed. After the plastic filling and cooling processes are completed, the injection molding machine will open the mold and the ejector pins will be pushed by the ejector plate, pushing the product out. The ejector plate compresses the mold's spring during the ejection operation. When the press returns to its initial position, the force acting on the

plate is no longer present. The spring compression force will now assist the plate in returning to its original position. The return pins are involved in this operation.

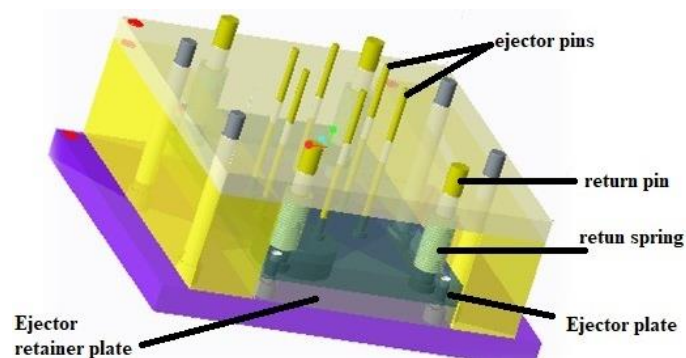


Figure 2.14: Ejector system

### 2.8.1. Ejector pins

Proper positioning of the ejector pin system will aid in the easy removal of the product from the mold, ensuring the product's external look. Ejector pin placement and distribution must apply a modest and consistent force across the molded part. Too few little pins, for example, will result in severe shear stress, causing the pins to punch holes in the molded object. Some suggestions for configuring the ejector system:

- Ensure the rigidity of the pins, avoid the case that the pin is too long but the diameter is too small.
- The distance between the ejector pins is consistent.
- The ejector distance should be at least 5 mm greater than the product's height.

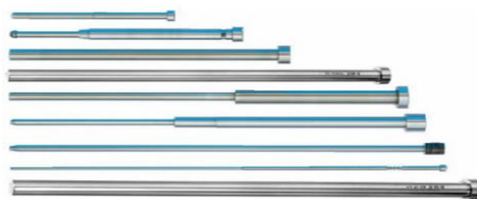


Figure 2.15: Some ejector pins design

### 2.8.2. Other components

- The ejector plate moves with the ejector system to push the product out of the mold cavity. And is placed on the bottom clamp plate (rear clamp plate).
- The return pin system has the effect of bringing the ejector plates back and keeping the ejector plate in place while the mold is completely closed. For many simple mold systems, a springless return pin can be used.

- The retainer plate keeps the push pins, ejector plate and return pins in place at all times.



# 3. DESIGN, SIMULATION AND ANALYSIS OF PLASTIC PRODUCT

## 3.1.Overview

Following the development trend of society, more and more plastic products are born to replace other materials. Especially plastic products for food and beverage processing needs. The Plastic Ice Tray is designed based on the convenience of use in families, beverage stalls, or restaurants.

According to its practical needs, the Ice Tray is designed on the following basis:

- Small, compact, lightweight can be used anywhere, any surface.
- Smooth surfaces.
- Withstands cold temperatures.
- Non-toxic and easy to recycle.

The product is designed using the NX 11 software application.

## 3.2.Product design requirements



Vietnamese-German University

Technical requirements of the product:

- The product has the right shape and size according to the design.
- The inner surface of the product achieves high gloss.
- The product is light and convenient to use, resistant to impact.
- Use recyclable materials.

Aesthetic requirements:

- The surfaces of the mold are smooth.
- The product has a clean look

## 3.3.Computer Aided Design and Computer Aided Engineering

With the help of computer systems and control structures, the calculation, design, and processing of mechanical products have been turned into full automation in the current era.

Software systems are increasingly rich with many different applications to meet specific types of work. With the help of a computer system equipped with CAD/CAE software, it is much easier to design and manufacture highly complex mechanical products. In order to improve the efficiency of production, software manufacturers are always trying to improve their software packages and extension modules. In the field of mold making, one of them is the Mold Wizard module, integrated into the Unigraphics NX software. Because of the outstanding features and features of this module, they can help

- Reduced design and fabrication time of ancillary components.
- Reduce costs for mold design and manufacturing.
- The parts are manufactured according to certain standards to ensure high accuracy.
- Improve productivity and economic efficiency.

In the following section, the program NX 11 was studied from scratch and used for designing the part of the product in this thesis. Melt flow simulation and analysis were carried out in Easy Fill Advance, while Mold Wizard was the main tool for mold base designing.

### **3.3.1. NX 11**

For engineers who need to deliver creative products in a collaborative setting, NX 11 for Design is an integrated product design solution that streamlines and speeds the product development process. In an open, collaborative environment, NX provides the maximum level of integration amongst development disciplines. It is appropriate for 2D and 3D design of a required part with many characteristics and interfaces, as well as part design assembly for a mold aggregate compilation. NX 11's finest features are automated wizards that aid with built-in modeling, such as holes and cuts with uniform size reductions. Finally, it offers a dynamic and diverse set of possibilities for building assembly structures out of pieces and linking the elements in the image to the assembly.

### **3.3.2. Easy Fill Advance**

From simulation and analysis of a melt flow by selecting the injection point and optimizing it for optimal phase to filling cavity values, clamping force, and filling time

analysis, NX 11 Easy Fill Advance can analyze the ideal phase of injection parameters under different mold design parameters such as clamp force, temperature, and pressure.

### 3.3.3. NX Mold Wizard

The Mold Wizard allows for rapid reaction to design modifications as well as high-quality molds. Mold Wizard has the following advantages:

- Using enhanced capabilities, you may speed up and simplify the design of the most difficult molds.
- Delivering levels of productivity that far beat traditional CAD applications.
- Integrating complex elements of design technology into automated sequences.
- Including intricate design technologies in automated sequences, extensive data import, and the use of translators or native-format readers to import design files from outside sources.

It helps simplify the core and cavity geometry design process by utilizing NX's automated functions, allowing us to: apply automated shutoffs and parting of core and cavity based on manufacturability concerns; design parting surface rapidly and associatively; divide mold to allow complicated slider designs; and automatically evaluate the core and cavity design for external interference.

For both prototype and production-scale multi-cavity molds, the NX Mold Wizard can help with the layout of the cavity, core, components, and mold base. Configure the mold using standard part libraries, component systems, and mold bases. Insert parametric cooling channels and instantly add and trim ejector pins. Gates, runners, pins, screws, and other basic components are included.

### 3.4. Product design using NX 11

The process of product design on NX 11 software is carried out through the following steps:

- Step 1: In the **Menu**, select **Insert>Design Feature>Block** create a block, with the below parameters (see Figure 3.1).



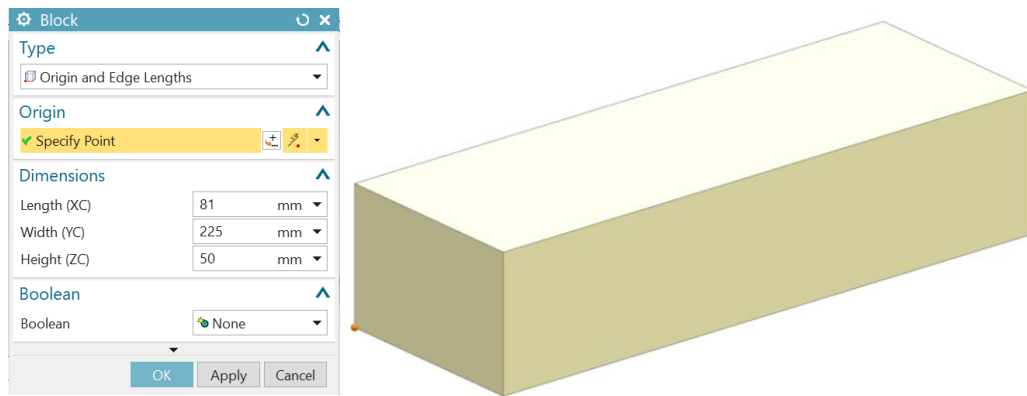


Figure 3.1: Create the block

- Step 2: On the block surface, using **Extrude** to create the concave of the Ice Tray with 30 mm depth and 10 degree of draft angle.

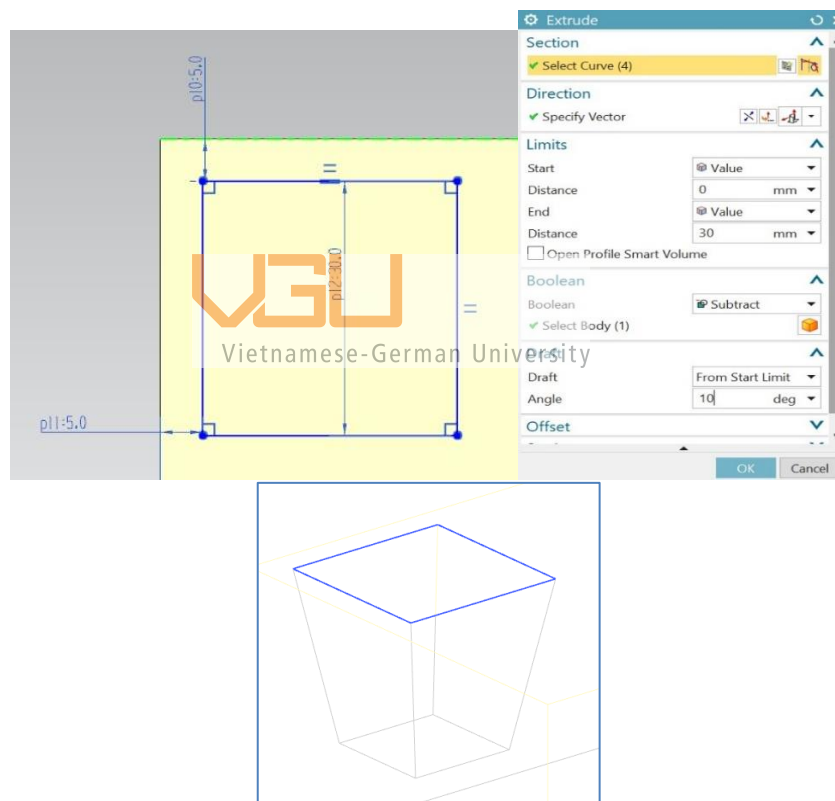


Figure 3.2: Create the concave of the Ice Tray

- Step 3: Using **Edge Blend** command to blend all the sharp edges around the concave.

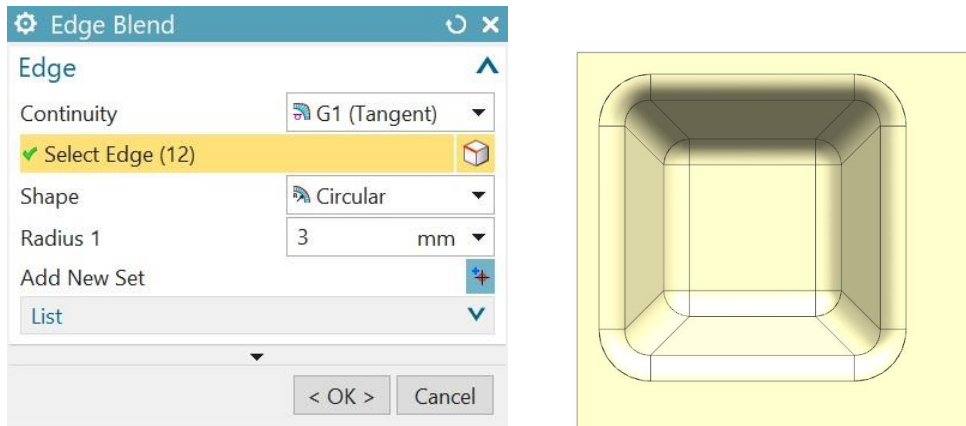


Figure 3.3: Edge blending

- Step 4: Using **Pattern**, select the Extrude and Edge Blend features for step 2 and 3 to create a pattern (see Figure)

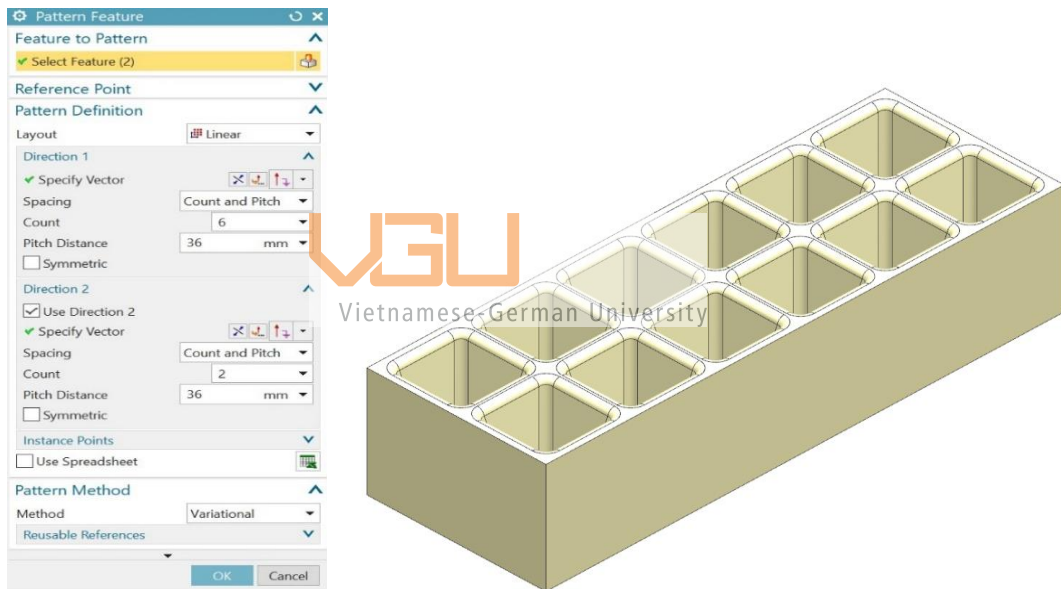


Figure 3.4: Create pattern

- Step 5: Form the **Menu**, choose **Shell** command and select the whole part to create the Ice Tray's shell with 1.8 mm thickness.

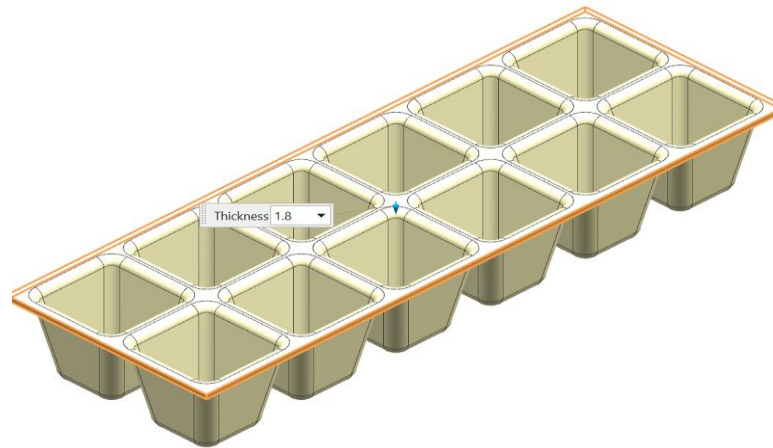


Figure 3.5: Create the shell

- Step 6: Continue with the **Edge Blend** command to blend all other sharp edges around the Ice Tray.

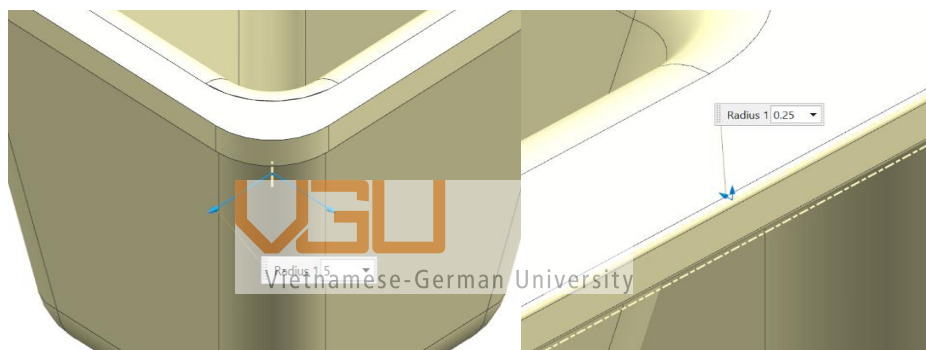


Figure 3.6: Edge blending the rim

- Step 7: Form underneath the tray, create a **Datum Plane** with offset 5 mm (see Figure). While on the new plane, create a **Sketch** between the midpoint of each concave of the tray as the base of the ribs.

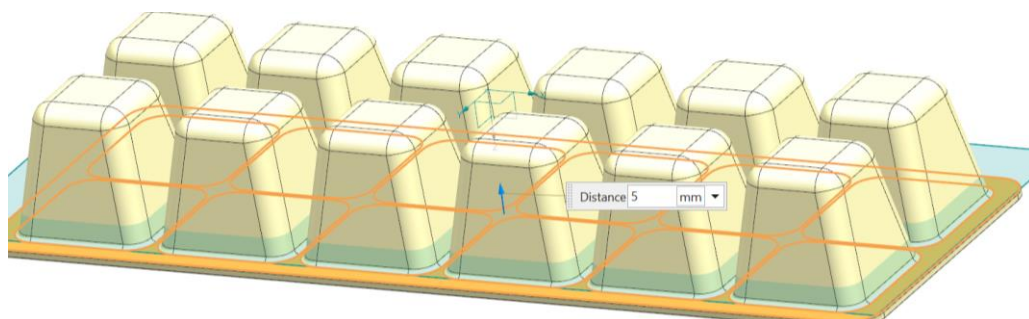


Figure 3.7: Create datum plane

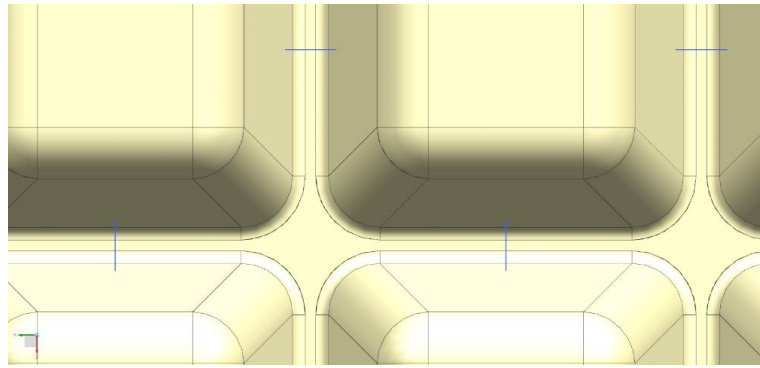


Figure 3.8: Sketch the base of the ribs

- Step 8: Select the **Rib** command with the target body as the Ice Tray and the rib section as the sketch from step 7 to create ribs with 1.8 mm thickness.

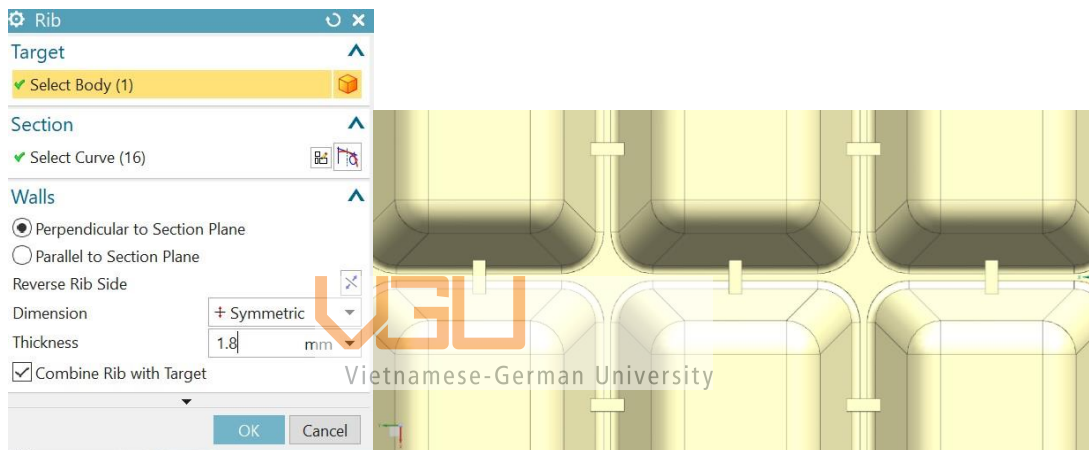


Figure 3.9: Create the ribs

- Step 9: Once again, using **Edge Blend** to blend all the sharp edges on the ribs.

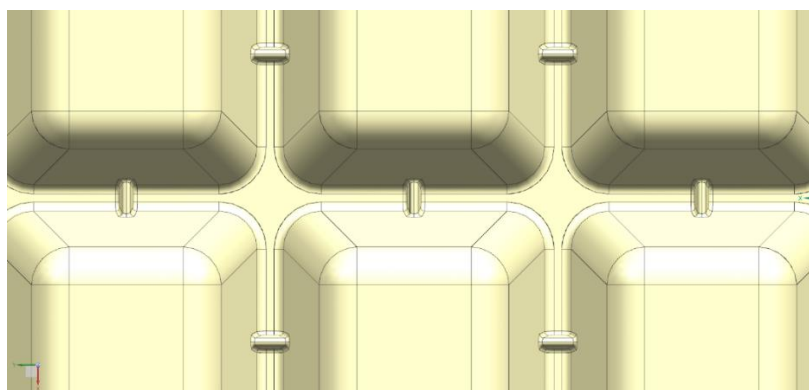


Figure 3.10: Edge blending the ribs

- Final product:

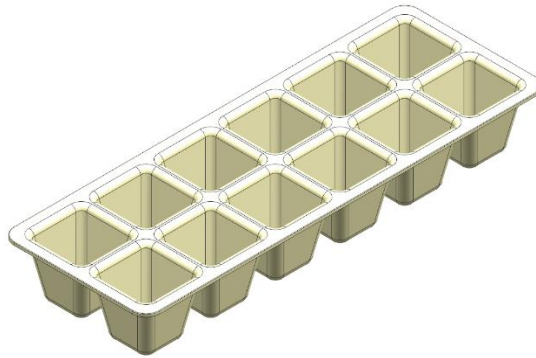


Figure 3.11: Final Ice Tray product

### **3.5. Material flow analysis using NX 11 Easy Fill Advance**

#### **3.5.1. Overview**

The injection molding process of plastic products is a complex process and is governed by many factors. After designing the product on NX 11 software, the product needs to be analyzed using CAE (Computer Aided Engineering) software before going into manufacturing a complete set of molds for the product.

In this part, the product will be analyzed on NX 11 Easy Fill Advance software to simulate the injection molding process, including:

Vietnamese-German University

- Filling process.
- Cooling process.
- Forming defects process.

Through the above analysis results to draw:

- Reasonable gate location.
- Simulate the process of filling plastic into the mold cavity when injection.
- Simulate defects that occur in the injection molding process such as warping, porosity, welding line to find out how to fix the errors, reduce time and cost before putting into actual manufacturing.
- Define injection molding parameters such as pressure, temperature, clamping force, filling time, and cooling.

#### **3.5.2. Initialize project**

The material of this Ice Tray is chosen to be PP. According to the official physical property table, the forming shrinkage of PP is around 0.019 to 0.021, so the average value 0.02 was chosen.

Following that, the general steps were followed, which included importing the file into NX Easy Fill Advance and meshing it using the approach stated above. After achieving the desired mesh statics, the melt flow methods are as follows:

- Material customizes: Generic PP
- Injection Pressure: 110 MPa
- Mold Temperature, 40 degrees Celsius
- Melt Temperature, 240 degrees Celsius
- Injection Molding Machine Advanced Settings:
  - Maximum Machine Injection pressure: 220 MPa
  - Machine Clamp Force: 100 tones

### 3.5.3. Best gate locations analysis

The best gate position should ensure uniform filling in all directions. The increased number of gates will reduce the time of plastic injection but make the design and machining process more complicated. It will also increase the possibility of generating weld lines and causing traces at the injection site of plastic on the part and loss of product aesthetics.

The following section of simulation and analysis of the material flow using NX Easy Fill Advance can be determined by these steps:

- Step 1: In the **Easy Fill Advance** menu, select the cavity of the simulated part. Then select the material setting as Polypropylene.

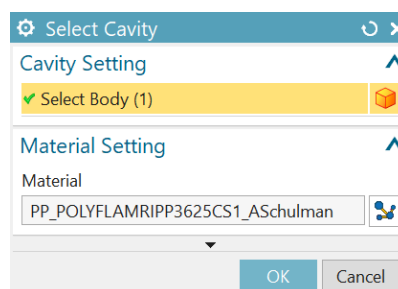


Figure 3.12: Select Cavity and Material Setting

- Step 2: In the **Gate Indicator** option, the number of gates required for this molded part can be determined by calculating the Flow length/Thickness Ratio (L/t ratio) to measure the Moldability (how easy the part can be molded).

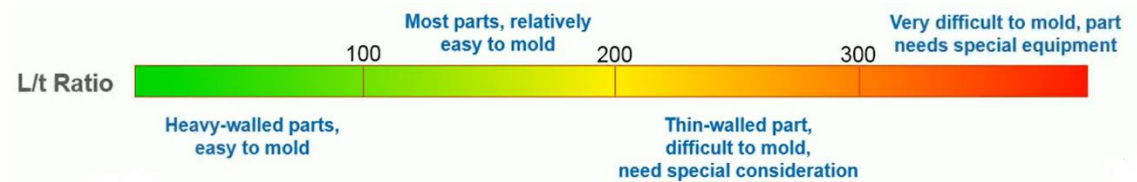


Figure 3.13: Flow length/Thickness Ratio spectrum

Figure 3.14 shows that the moldability of the product is affected by the range of L/t ratio values. From the indicator, it is clear that this product can be molded with just one gate since the L/t ratio is less than 100. However, for the size of this product, one gate will take too long for filling time and could cause other major defects. The question is: is it possible with two gates?



Figure 3.14: One gate L/t ratio

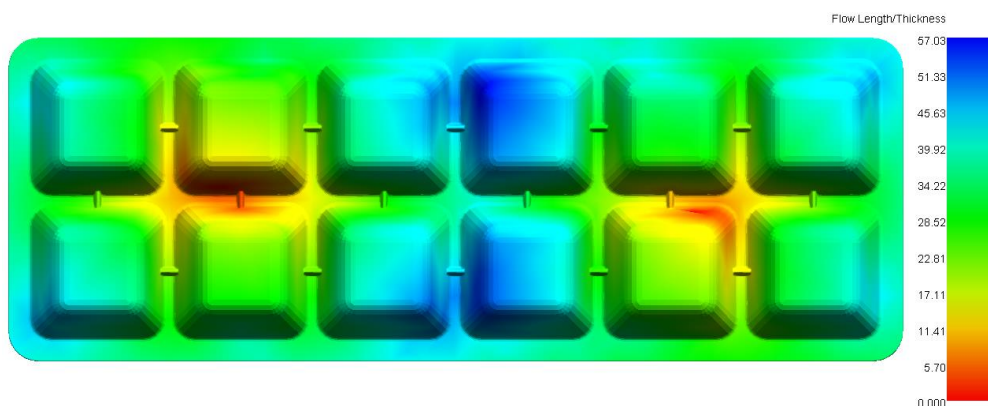


Figure 3.15: Two gates L/t ratio

From Figure 3.15, it is clear that by adding two gates, the L/t ratio also drops, so it is possible to add gates at the indicated area of the product, which is colored red. Unfortunately, the automatic gate location recommended by NX Easy Fill Advanced is not reasonable for the manufacturing process, as can be seen in Figure 3.16, so the gates have to be manually added and the L/t ratio must be recalculated.

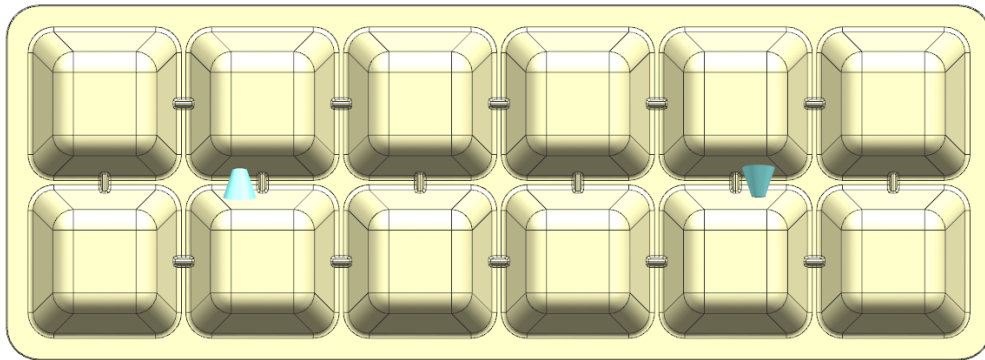


Figure 3.16: NX Easy Fill Advanced automatic gate location recommendation

- Step 3: Use the **Create gate** option, select the type and size of the desired gate. New gate locations should be added near the ideal location, offset to the edges near the parting surface. In this case, the edge gate design was chosen. Also noted that always using the same gate type for one product.

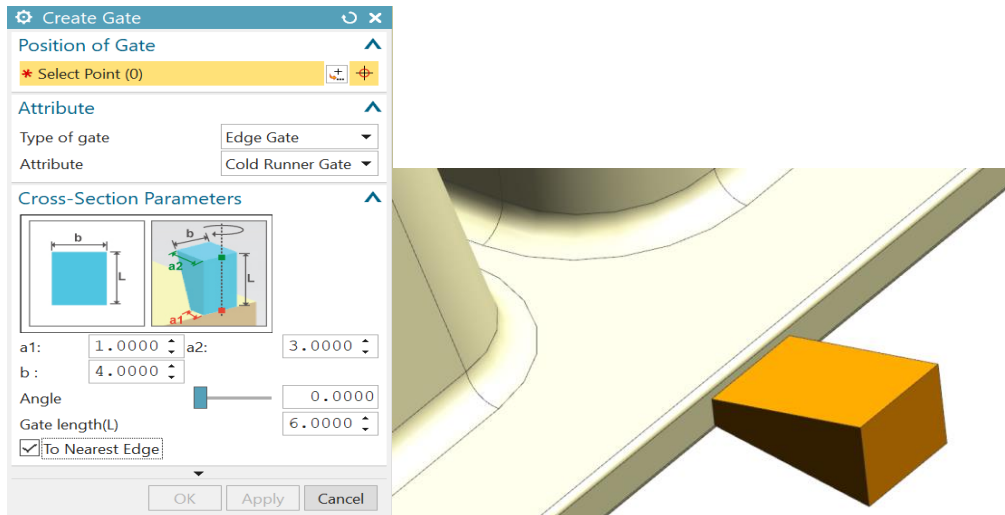


Figure 3.17: Gate design



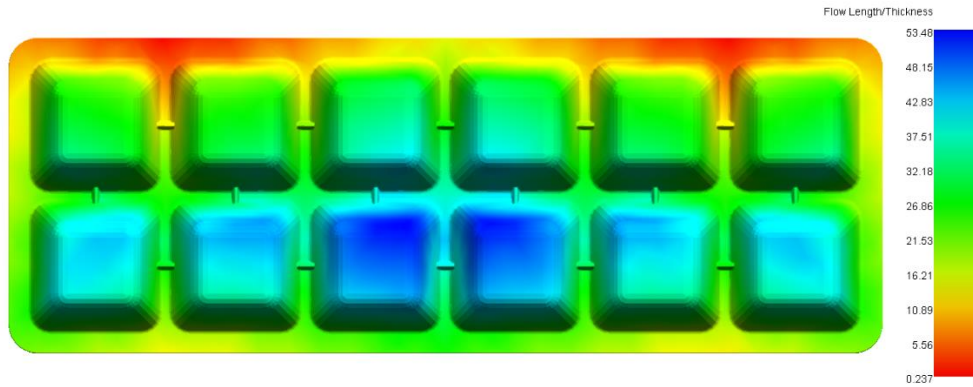


Figure 3.18: Recalculate L/t ratio of new gates location

From Figure 3.18, it is shown that the new gate locations have a sufficient L/t ratio, which means it is fine to keep this design.

### 3.5.4. Feeding system design

In this thesis, the number of cavities in this mold was chosen to be two, and the sprue location was selected to be the coordinate system position of the layout.

The density of the chosen material, PP, is  $0.915 \text{ g/cm}^3$  (Wang, Chang, & Hsu, 2018).

NX analysis shows that the volume of a single plastic part is

$$V \approx 71.71 \text{ cm}^3 \quad (3.1)$$

A single plastic part's mass:

$$M = 71.71 \times 0.915 = 65.62 \text{ g} \quad (3.2)$$

From the NX 3D environment, the distance between the sprue location and any nearest gate is estimated to be 70 mm. Therefore, the runner diameter can be calculated using equation 2.1 as:

$$D = \frac{65.62^{\frac{1}{2}} \cdot 70^{\frac{1}{4}}}{3.7} = 6.33 \text{ mm} \approx 6 \text{ mm}$$

Then, in the Easy Fill Advance menu, choose Runner and sketch the runner channel connecting all four gates together on the parting surface. Note that the distance from the sprue premeditated location must be equal in order to obtain a balanced material flow.

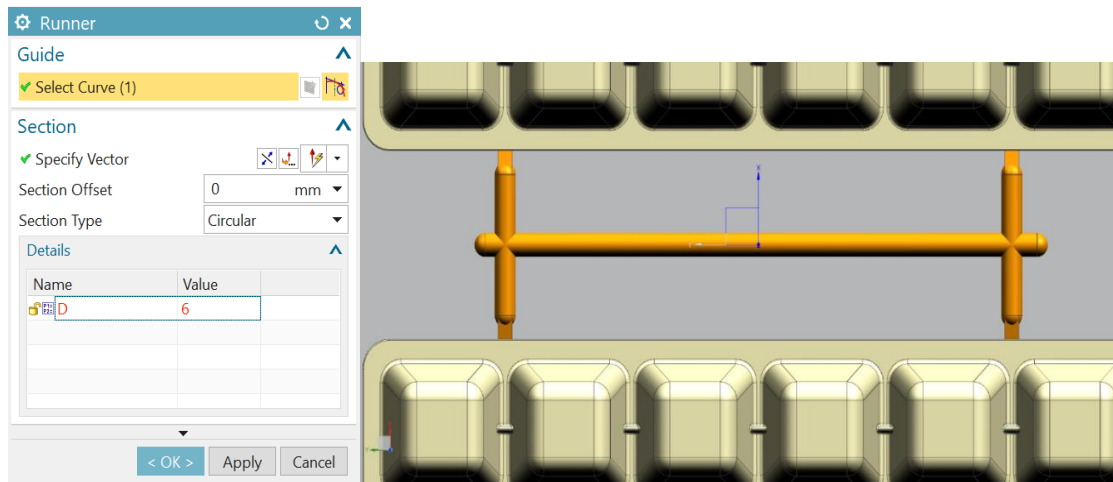


Figure 3.19: Runner design

Finally, the sprue design was created with the following parameters in Figure 3.20.

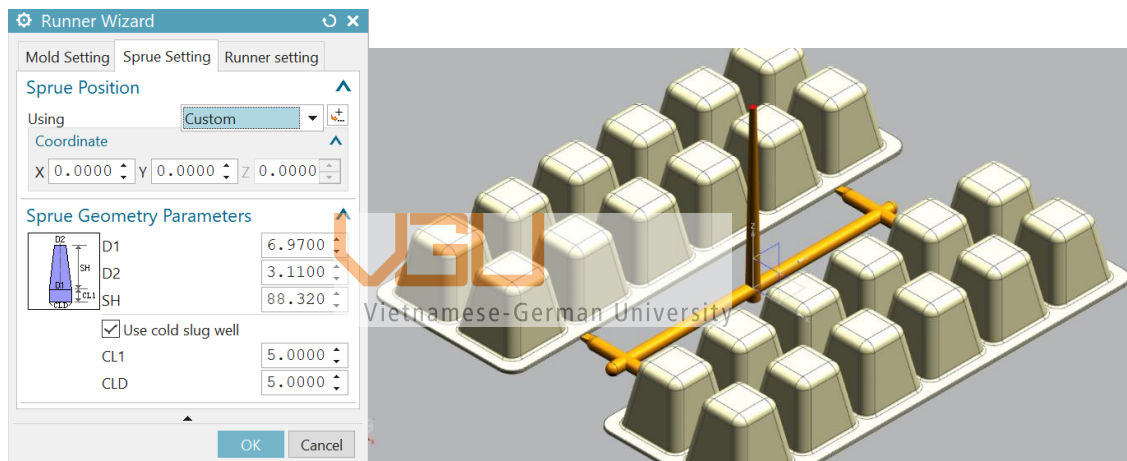


Figure 3.20: Sprue design

### 3.5.5. Filling rate analysis

Filling time depends on other parameters of the pressing process, including pressure, runner size and shape, gate size and shape, and injection molding process temperature. The shorter the filling time, the higher the productivity, reducing the possibility of defects.

The results of the fill-in analysis are presented in Figure 3.21. The analysis results show that the filling time at the furthest positions on the product is 3.029 seconds. The filling process is represented by the colors gradually changing from red to blue as the filling time increases. Notice that, as soon as the injection molding process starts, the molten plastic is filled into the sprue first. After that, liquid plastic gradually fills the

runner and then the part of the product closest to the gate, then fills it gradually in the direction of increase and finally the product rim. In other words, plastic is preferred to fill in the area closer to the mouth of the gate. After this area is filled, the resin is pressed into the rest. The blue areas are filled in last. The time scale shows that the entire cavity filling process takes 3.029 s.

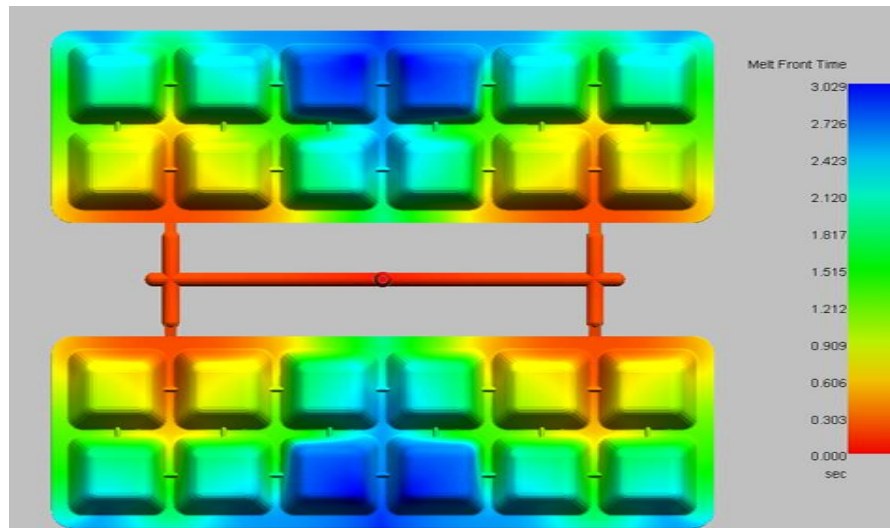


Figure 3.21: Melt Front Time analysis

### 3.5.6. Temperature analysis

Vietnamese-German University

Mold temperature is an important issue. If the mold temperature is too high, it will affect the solidification of the plastic. The plastic is difficult to freeze, leading to many problems, such as having to wait for a long injection molding process. For the product to solidify completely and at the same time, to increase the cooling time of the mold, which requires the design of the cooling system and cooling materials.

On the contrary, if the mold temperature is too low, it will cause the plastic to easily solidify, making it difficult to fill and easy to cause defective products. Solving this problem is also complicated because it will involve cooling issues, machine pressure, plastic conduction system, and plastic process temperature.

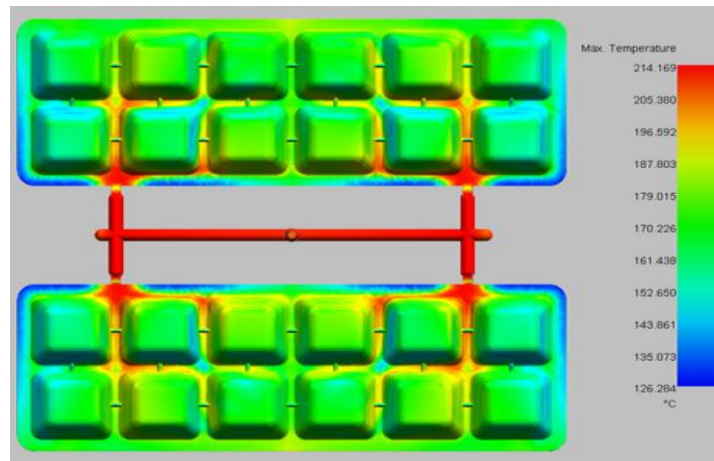


Figure 3.22: Temperature analysis

The result showed that the temperature at the top of the flow is about 214 degrees Celsius, which is equal to the melting point of the material at 210 degrees Celsius. However, the temperature will gradually decrease to about 126.3 degrees Celsius.

The temperature of the outer part of the material due to contact with the mold surface should lose heat quickly and solidify. The inner material remains at a high temperature. This is the cause of shear stress and frictional heat in the injection molding process.

### 3.5.7. Pressure analysis

Pressure affects the plastic flow rate in the mold, dimensional stability, and mechanical properties of the product. Low pressure will increase press time, causing uneven shrinkage on the product. High pressure will affect the clamping force holding the mold. Excessive pressure increases the possibility of burr generation at the parting surface and the position of the gap points.

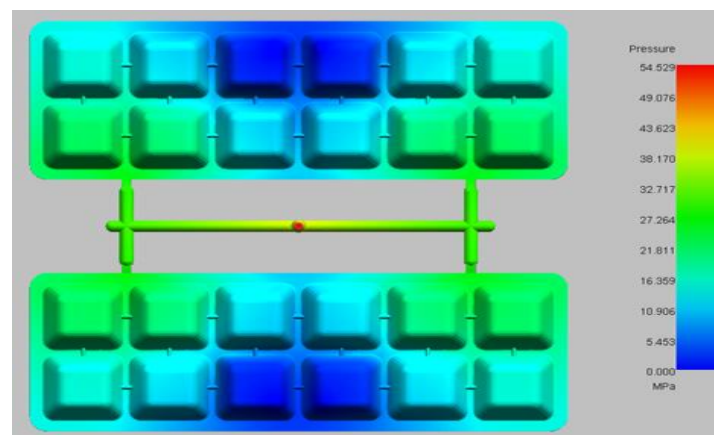


Figure 3.23: Pressure analysis

The pressure is unevenly distributed at those locations as a result of pressure drop along the flow of the plastic, making the material compression uneven. The further away from the nozzle, the lower the pressure.

The analysis results in Figure 3.27 also show that the maximum injection pressure during the injection process reaches 54.529 MPa at the sprue tip and decreases gradually with the flow of the resin. Attention should be paid to this parameter when setting up the injection molding machine. Determining the correct mold pressure ensures the filling process of the mold cavity, helps us to control the stability of the product, control the mold filling capacity and material compaction, and the product quality will improve.

### **3.5.8. Clamping force analysis**

The clamping force is the force that keeps the two halves of the mold closed when the molten plastic flows into the mold cavity. The clamping force is increased to its maximum during pressurization and is maintained until the product is opened.

Clamping force is a factor affecting the machine capacity and the life of the mold. When closing the mold, the large clamping force will cause a number of problems, such as: consuming machine energy; causing the surface of the parting part to collide strongly, which will cause surface blistering to wear out the mold and machine; reducing mold air release; and can prolong the pressing cycle. Therefore, in the process of setting the clamping force of the mold, it is necessary to minimize the clamping force. However, during the injection molding process, the pressure in the mold cavity will gradually increase to push the two halves of the mold out. If the clamping force is not enough, the product will be broken, reducing product quality.

The analysis results give a graph of the magnitude distribution of the clamping force over time. Mold clamping force during injection increases relatively evenly, but near the end of the injection cycle it increases sharply.

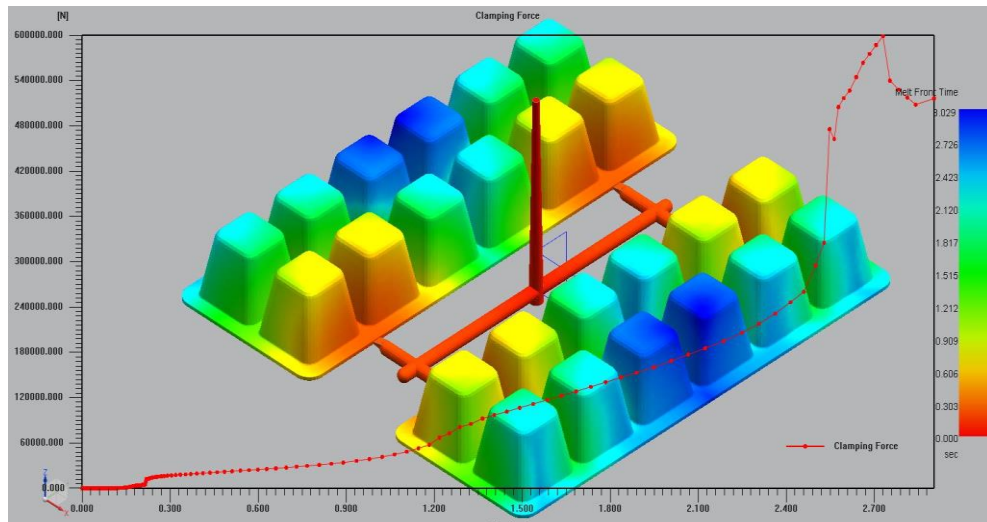


Figure 3.24: Clamp Force analysis

Mold clamping force process:

- In the first stage (roughly from 0s to 0.21s), the clamping force is relatively modest, essentially non-existent, and increases in an unstable manner. The plastic has only just begun to be pumped into the mold cavity at this point.
- Between 0.21s and 2.46s, the clamping force gradually and steadily increases from 1.5 tons to 25.08 tons; at this point, the plastic is being loaded into the mold.
- The clamping force peaks at 59.86 tons at the end of the injection cycle in the final stage (from 2.73s until the completion of the filling). At this stage, the force develops rapidly because, at the conclusion of the filling process, a period of clamping the mold is required to force the fill material flow (compensate for shrinking material), thus the compression pressure is very high, and the clamping force is strong.

Optimum clamping force can be determined through a number of tests. The estimated mold clamping force can be used as the starting point. If in the process of pressing the mold to create a burr, it is necessary to increase the clamping force on the mold until the mold runs well, then there is no need to reduce the clamping force of the mold anymore. Otherwise, slowly reduce until the burr appears and then increase the mold force by 10–20% to make sure the product is pressed well.

Note that the clamping force should not be more than 80% of the maximum use limit of the machine; the rest is for safety. Therefore, it is necessary to pay attention to setting the corresponding clamping force for the plastic injection machine.

### **3.5.9. Defection analysis**

#### **3.5.9.1. Air trap**

Air trap defects on products appear usually due to variable reasons:

- When products have concentrated streams, they often collect air in one spot causing air bubbles in that spot.
- Often air traps are caused by unbalanced fill lines.
- During the mold filling process, air is trapped in the product in areas near the surface of the product.

The analysis will help determine the location of the air trap concentration on the product, thereby offering alternative solutions to minimize the occurrence of porosity defects on the product to avoid affecting the mechanical properties as well as the aesthetics of the product.



Vietnamese-German University

The analysis results show that the air trap is concentrated primarily at the product's ribs and edges, where the product thickness abruptly changes, with some gathering in the middle, where the flow is obstructed and changes. The air is trapped and unable to escape at these points, as the plastic flows encircle the air and create air pockets.

To remedy the defect, the air outlet grooves should be placed near the points where the air is concentrated, and at the same time, calculation should be made to reduce the injection speed to a reasonable level, because if injected at a high speed, the air bubbles cannot escape. However, it is necessary to prioritize the optimization of the vent system before taking into account the reduction of the injection speed.

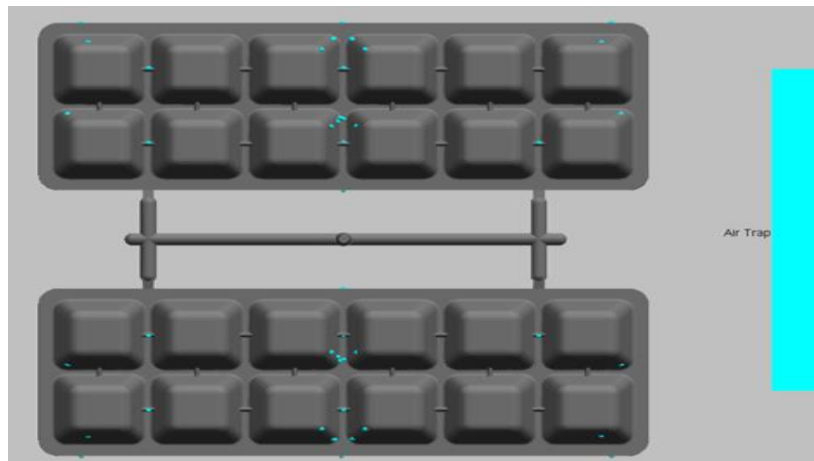


Figure 3.25: Air trap analysis

### 3.5.9.2. Weld lines

Weld lines appear as black marks at the end of the flow (air trapped), V-shaped marks, or lines of different colors, especially when using inorganic colors. Conspicuous in dark or clear products with smooth surfaces, the weld lines can seriously affect the mechanical properties as well as the aesthetics of the product.

It is shown in Figure 3.26; weld lines appear due to the meeting of plastic flows. When flowing through joints and walls, the flow is dispersed, but due to the heat loss process, when the flows mix, there is no connection, making the product after injection molding easy to break. Weld lines seriously affect the mechanical properties as well as the aesthetics of the product.

To fix it, there are a few measures:

- Increase melt temperature, injection rate or pressure.
- Expand the runner channels and gate size.
- Reposition gates for easier flow.
- Increase mold temperature and melt temperature.



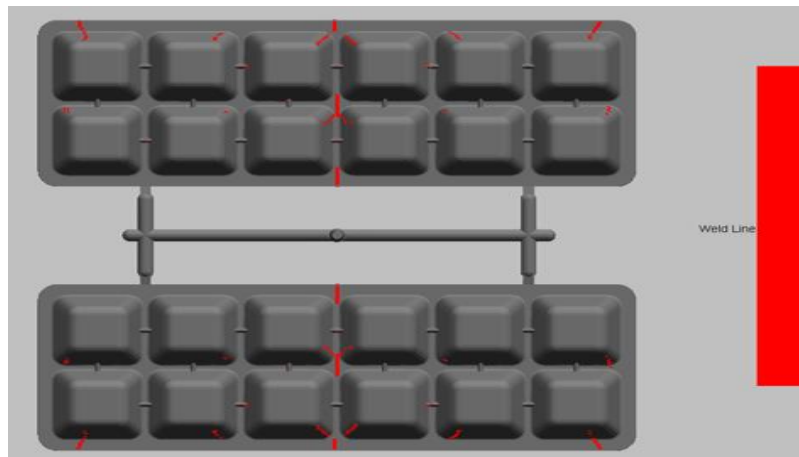


Figure 3.26: Weld Line analysis

### 3.5.9.3. Volumetric shrinkage analysis

Under the influence of temperature, the material expands in the injection molding process. After the process of pressurizing, the temperature in the mold gradually decreases to the opening temperature of the mold. The product tends to shrink.

Product warping is caused by uneven cooling. Residual stress causes the product to bend in different directions due to different product thicknesses.

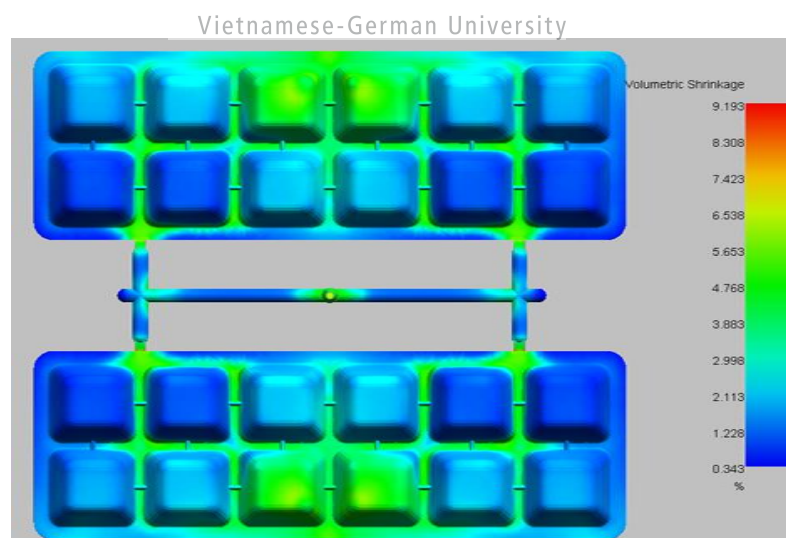


Figure 3.27: Volumetric Shrinkage analysis

Based on the percentage volume shrinkage chart, it can be seen that the volume of shrinkage lies mostly, ranging from 0.343% to 9.193% compared to the original volume. Shrinkage area from 7% to 9.193% is small and almost negligible on the product. Shrinkage areas of 1% or less are concentrated mainly on the outer edge, so it has little effect on product quality. Shrinkage area is mainly distributed from 1% to

6.538%. The shrinkage density is also unevenly distributed over the parts of the product, which ensures that the product after shrinkage has an uneven, distorted shape.

The reason for this is that there is an uneven shrinkage between the body and the edge of the product because of the different thickness between the two products. Therefore, this shrinkage is inevitable due to design requirements to ensure the aesthetics and usability requirements of the product.

Material shrinkage greatly affects the shape formation and the aesthetic factor of the product. Knowing this shrinkage, we can adjust the amount of material compensated in the cooling and shrinking process appropriately.

### 3.5.10. Analytical summary table

The results after calculation have been evaluated with the proposed designs being suitable and can be processed. The product after injection molding has the shape and size within the allowable range of the manufacturer's designed products.

The summary table is presented below.

Item Name	Item Data
<b>Filling time</b>	3.029 seconds
<b>Max. Melt temperature</b>	210 degrees Celsius
<b>Min. Melt temperature</b>	126.3 degrees Celsius
<b>Max. Injection Pressure</b>	54.529 MPa
<b>Max. Clamping force</b>	59.86 tons

Table 3.1: Summary table

## 4. STRUCTURAL DESIGN OF INJECTION MOLD

### 4.1. Initialize project

Before starting the design process, the molded product needs to go through several steps of checking. First is the checking of the draft angle of the product relative to the mold separation direction in order to see if it is reasonable compared to the direction of mold separation. When splitting the mold, the draft angle should never be zero, it should be at least 1-3 degrees from the parting surface so that when demolding, the product does not stick to the mold or break.

- Go to **Menu >Analysis >Molded Part Validation >Check Regions**. Here shown a dialog box saying **Check regions**:
- Select the **Face** tab and set the **Draft Angle Limit** to the desired minimum **Draft angle value**. Set the colors that correlate with the thickness in the table. Select **Set Color of All Faces**.

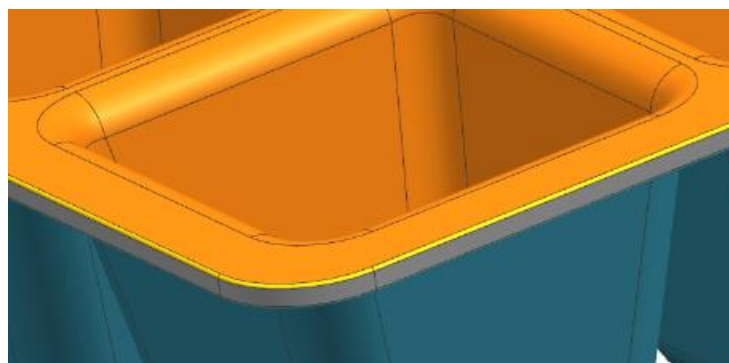
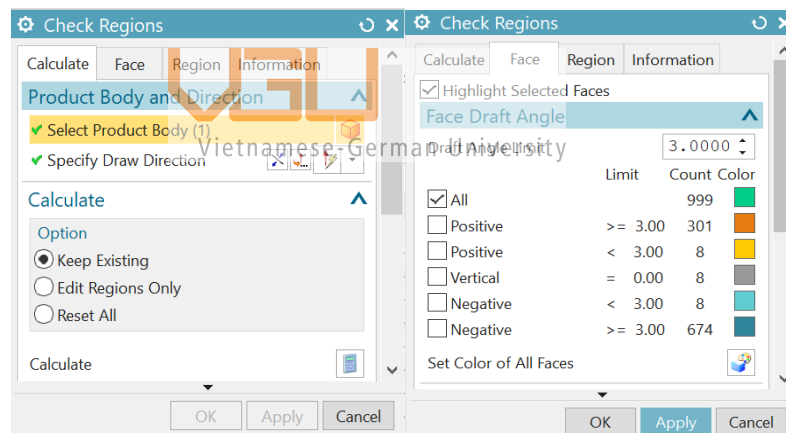


Figure 4.1: Check Region and Draft Angle Limit

Result: The surfaces display the color corresponding to the draft angle value on the **Check Regions** table. As shown in Figure 4.1, the only vertical surfaces (colored gray)

are located on the outer edges, so the product needed to be redesigned to have a new roughly 1-degree draft angle there.

Next is checking the thickness of the part. The purpose is to know the structure as well as the thickness of the detailed object to design runner channels, cooling channels, etc., in the most optimal way. Usually the thickness of the product is uniform so that the cooling is most uniform, and the material is poured evenly to avoid defects.

- Go to **Menu** > Go to **Analysis** > Select **Molded Part Validation** > Select **Check Wall Thickness**. On the **Check Wall Thickness** tab, select **Calculate Thickness**.

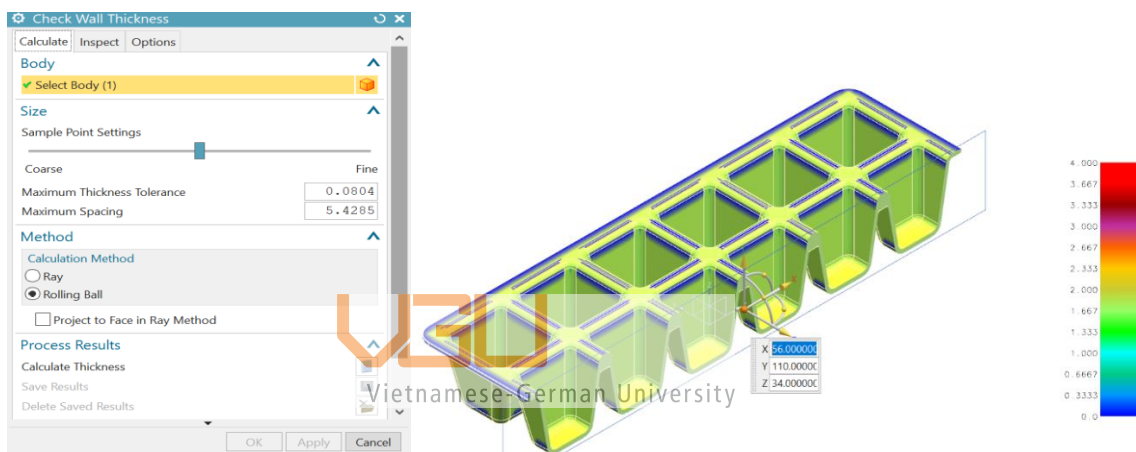


Figure 4.2: Check Wall Thickness

Result: The thickness of the faces corresponds to the colors on the right corner of the screen. From the analysis, it is clear that the product does not have any drastic changes in thickness throughout the whole part.

After completing the analysis and checking the structure of plastic products, the mold separation process is carried out through the following steps:

Step 1: To set initial parameters for details, select **Initialize Project**.

- Select the material for the product as PP, corresponding to a shrinkage of 1.02.
- Select configuration as Mold.V1.
- Select units of the Millimeter system.
- Click **OK**.

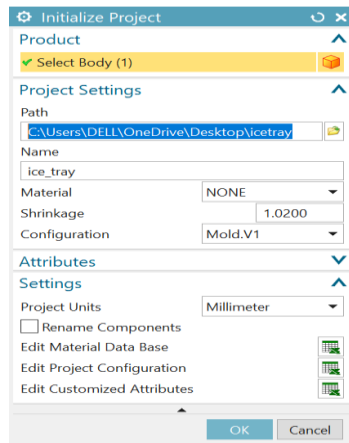


Figure 4.3: Initialize Project

Step 2: Preset the origin for all mold components. Select **Mold CSYS**.

- Check the **Current Mold CSYS**, select **Menu > Format > WCS > Dynamics**. Point the mouse to select the point on the desired main parting surface. Also rotating the coordinate system to find the suitable Z-direction.
- Right click on the desktop and click **OK**. Then the **Mold CSYS** dialog box appears, check **Lock Z-direction**, then check **Product Body Center** tab. Finally, click **Apply** and then click **OK**.

Vietnamese-German University

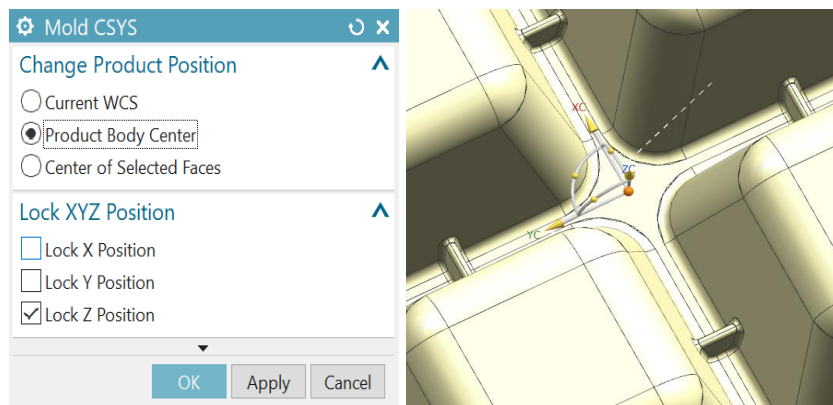
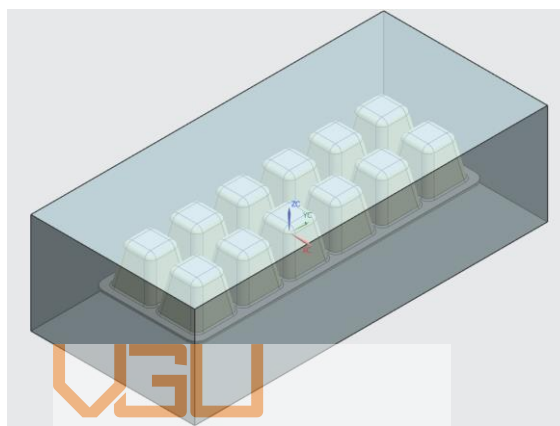
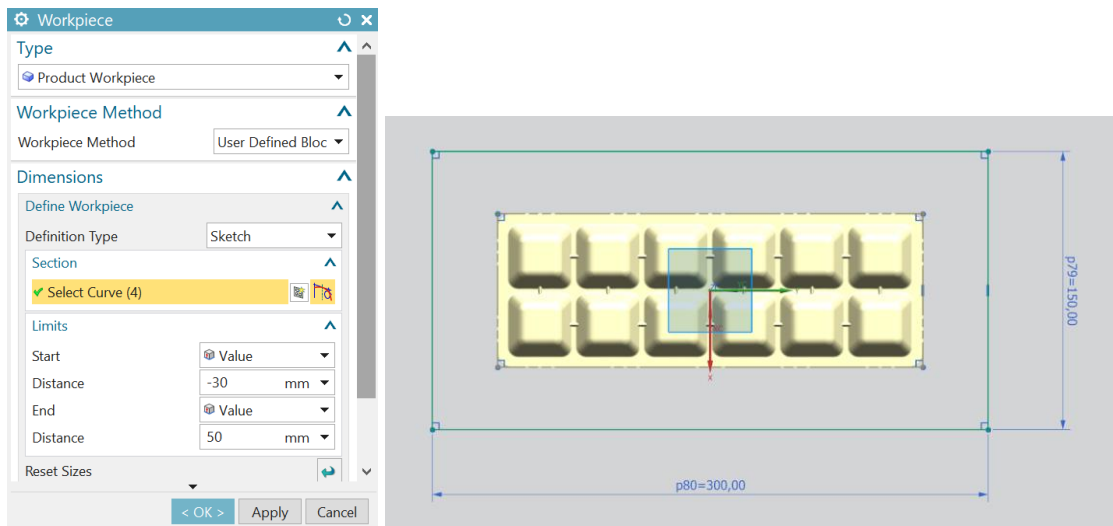


Figure 4.4: Setting coordinate system of the mold

Step 3: To create the workpiece for cavity and core creation, select **Workpiece**.

- Edit the size of the workpiece according to sketch, click on the **Sketch** section. The workpiece can be resized according to the Sketch profile with a size of 130x280. Click **Finish** (Ctrl + Q) after setting the desired size.
- Set the height of the workpiece in the Limits panel with the Sketch plane referenced. Here the height was set to start at -30 mm and extend to 50 mm.



Vietnamese-German University  
Figure 4.5: Create Workpiece

Finally, a 150x300x80 mm workpiece was created, covering the molded part.

Step 4: The next step is to arrange two products in one cavity using the **Cavity Layout** command. In this thesis, choose the parameters as shown in the figure below. The direction of choice is XC. After that, click **Start Layout** for NX to generate the 3D geometry of the desired workpiece layout and **Auto Center** to set the mold coordinate system in the center of the layout. To blend the corners of the layout, select **Insert Pocket**, choose R15 mm\_type 0.

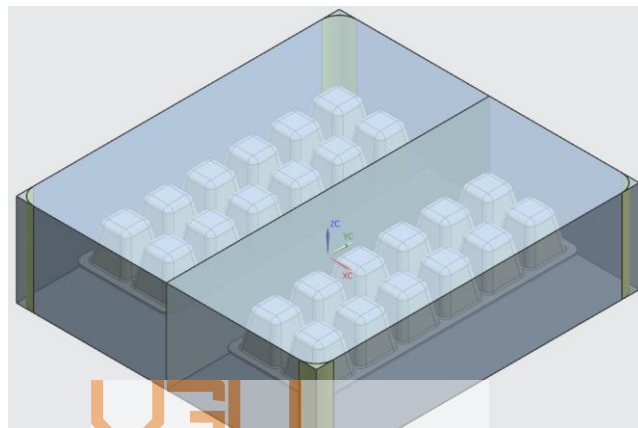
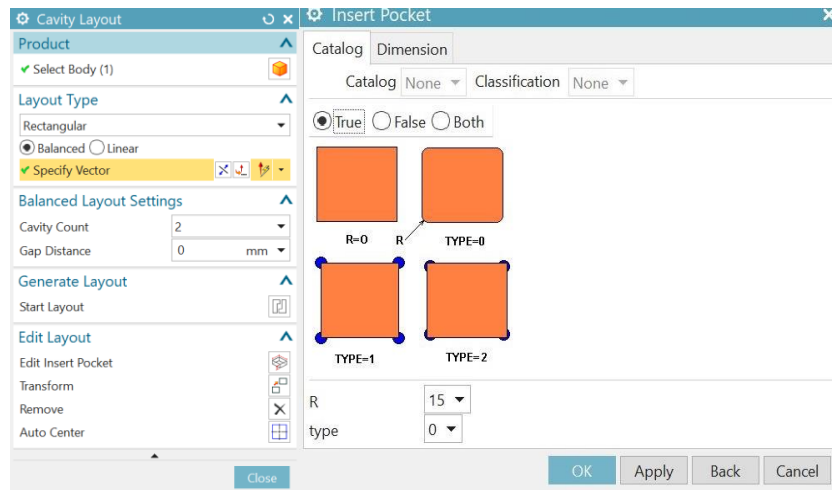


Figure 4.6: Setting Mold Layout

## 4.2. Parting Surface

Step 1: Locate the surfaces in the cavity and core sections of the mold.

- In the Mold Wizard menu, select **Check Regions**, then select the drawing direction for the part according to the Z axis. Then click **Calculator** to calculate.
- Next, determine the surface located on the cavity and core. Keep the results of the above table and switch to the **Region** tab. In the **Region** section, click **Set Region Color**. The product is set to 3 colors: **Orange** (Cavity), **Blue** (Core) and **Cyan** (Undefined Region).

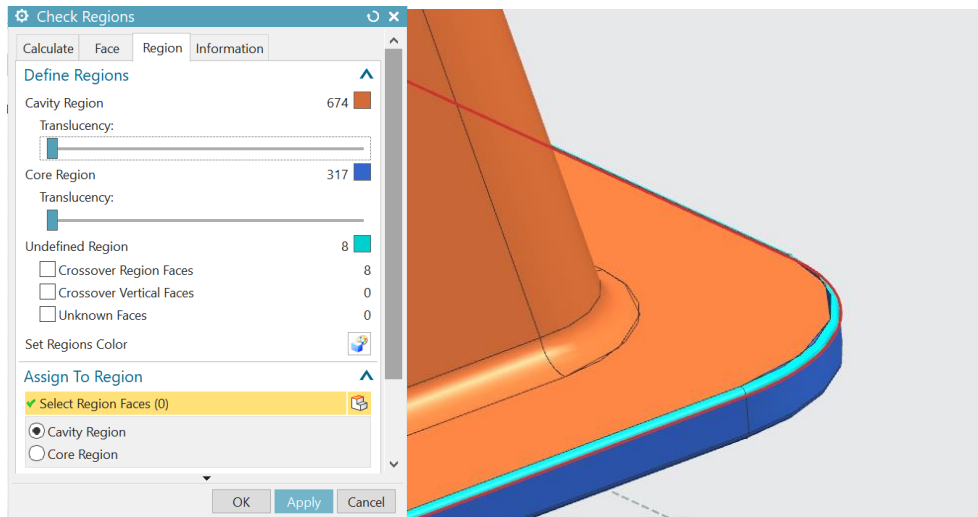


Figure 4.7: Identify the faces of the product to each region

- Next, press Shift + select the Undefined Region area to assign those faces to the Cavity area of the Mold, which will define those faces as a part of the cavity.
- Then click **Apply** in the **Check Region** dialog box. Finally, click **Cancel** to exit the **Check Region** dialog.

Step 2: Select **Define Regions** to assign the selected regions to the core and cavity parts of the mold by checking the **Create Regions** and **Create Parting Lines** checkboxes. From there, NX can automatically generate the parting surface of the mold.

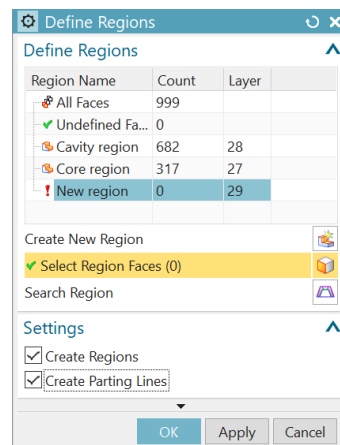


Figure 4.8: Define Regions

Step 3: Create the main parting surface using the **Defined Parting Surface** command. Since the selected product is quite simple, the design of the product's parting surface is also quite simple. Only one parting face is generated automatically by NX.



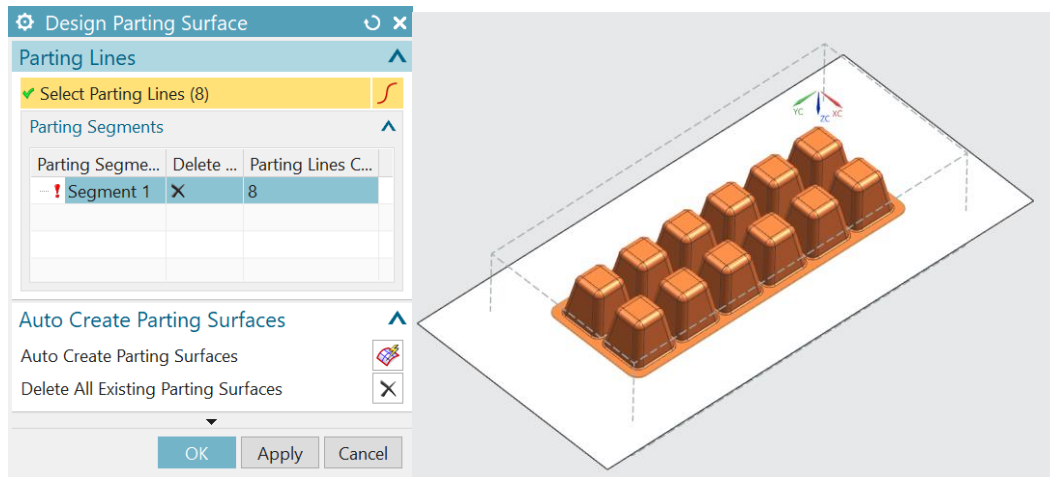


Figure 4.9: Create Parting Surface

### 4.3. Splitting Core and Cavity

Using **Define Cavity and Core**, the cavity and core molds can be divided into two parts.

- Click on the **Cavity region** to see all the product divisions.
- Click **Apply** to get the cavity mold.
- Then click **OK** to agree.

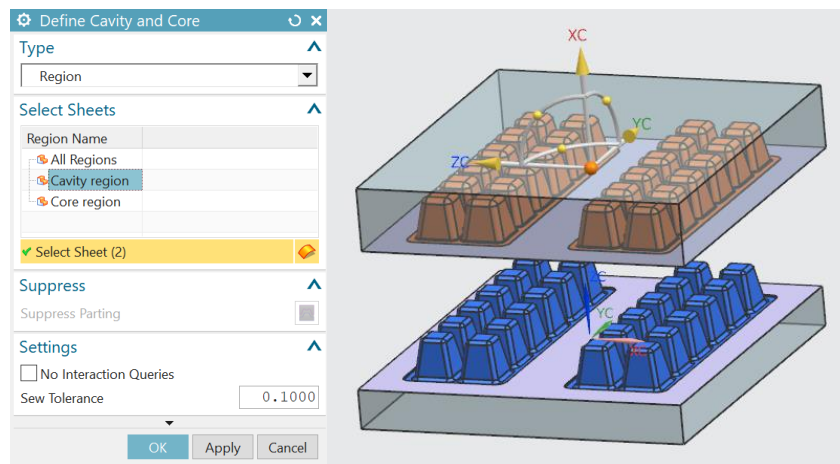


Figure 4.10: Cavity and Core splitting

Similarly, select **Core Region** and click **Apply** to get the core section.

### 4.4. Mold base selection

After the cavity and the core have been created, the next step in the mold design is to create the mold base.

Step 1: Click the **Mold Base Library**. In this thesis, the mold base temple FUTABA\_S, type SA, was chosen as the mold base. After that, the information window will show up with all the parameter information.

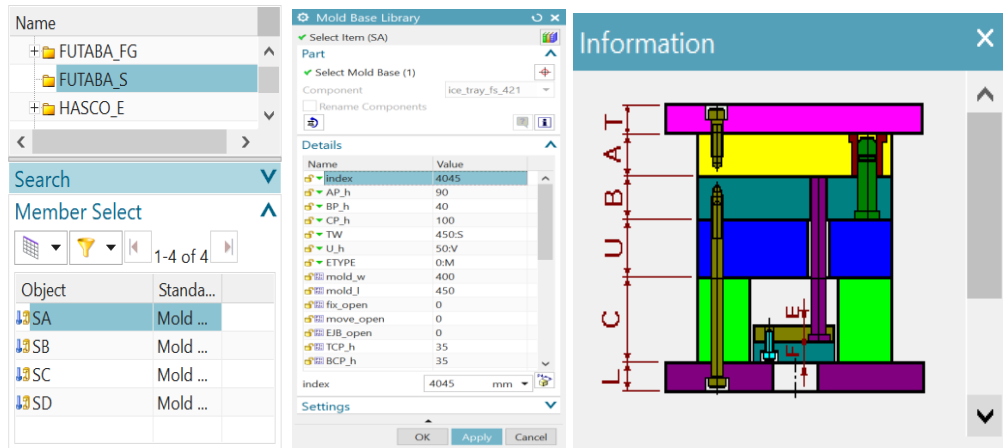


Figure 4.11: Mold base selection

Step 2: Adjusting the parameters of the mold base so that the size of the base is suitable for the product layout. Click **OK** to create the 3D geometry of the desired mold base.

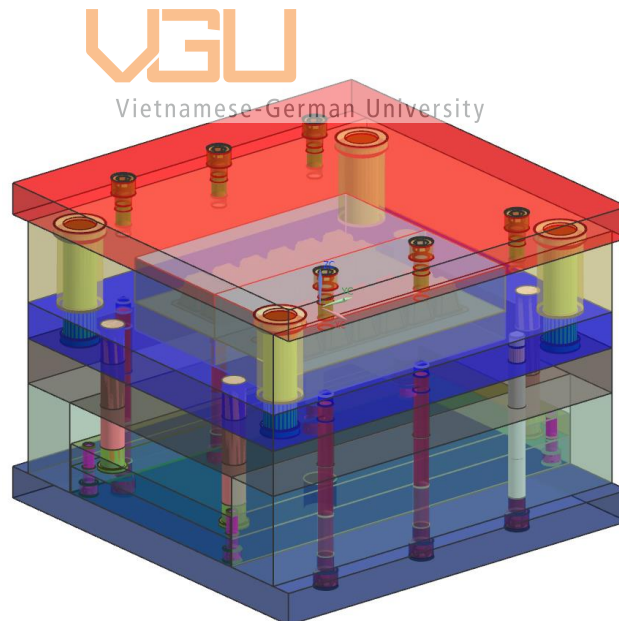


Figure 4.12: Mold base created

Step 3: The next thing is to insert the sprue bushing, with the DME sprue bushing DHR76 as the chosen bushing. Click on **Standard Part Library**, go to DME, go to Sprue bushing DHR76 and adjust the suitable parameters. When click **OK**, NX will automatically insert the parts in the middle of this mold tooling, so they will be created

in the coordinate system of the mold base plus Z. That's why it is very important when setting the center for the core and cavity layout.

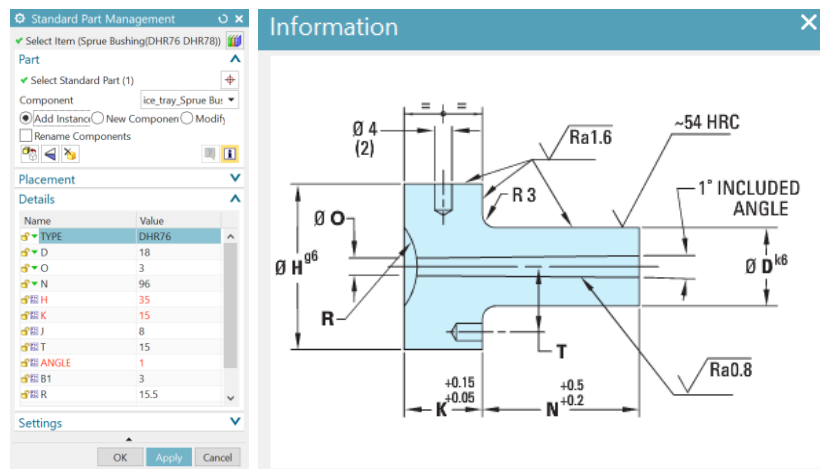


Figure 4.13: Sprue bushing design

Step 4: Next, insert the locating ring with mounting holes using the FUTABA standard. Same again in the **Standard Part Library**, go to the **FUTABA**, select **Locating Ring**, and choose **Locating Ring [M-LRJ]**. Based on the sprue bushing design, the locating ring is designed with the following parameters.

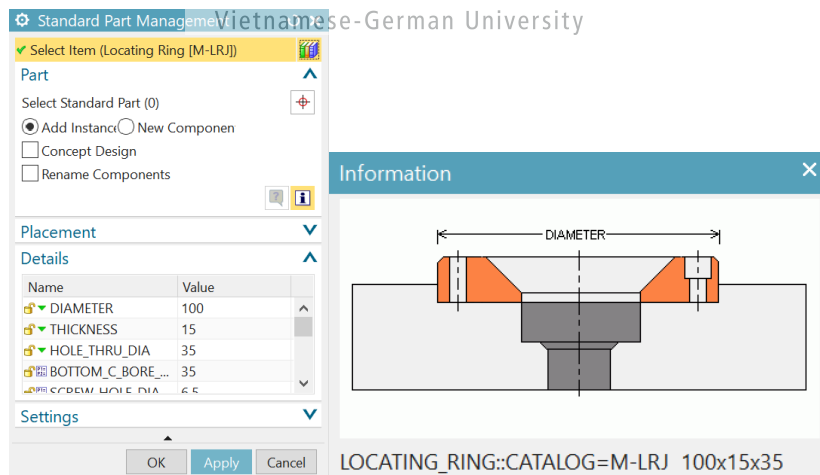


Figure 4.14: Locating ring design

#### 4.5. Cooling system

According to table 2.3, for a product that has a wall thickness of around 1.8 mm, the optimal size for its cooling channel diameter is  $d = 8$  mm. Following that, the distance between each cooling channel and the distance between the cooling channels to the product were chosen according to its recommended size, which is  $b = c = 2d = 16$  mm.

The next step is to create the pattern of the cooling channel, define the channel and cooling circuit, as well as generate plugs, etc. Due to the mold's relatively small size, the cooling channel is only placed on the cavity side of the mold in this thesis.

Step 1: To create the cooling channel inside the workpiece, in the **Cooling Tools** tabs of Mold Wizard, select **Pattern Channel** and sketch the cooling pattern on a plane that is 16 mm away from the product.

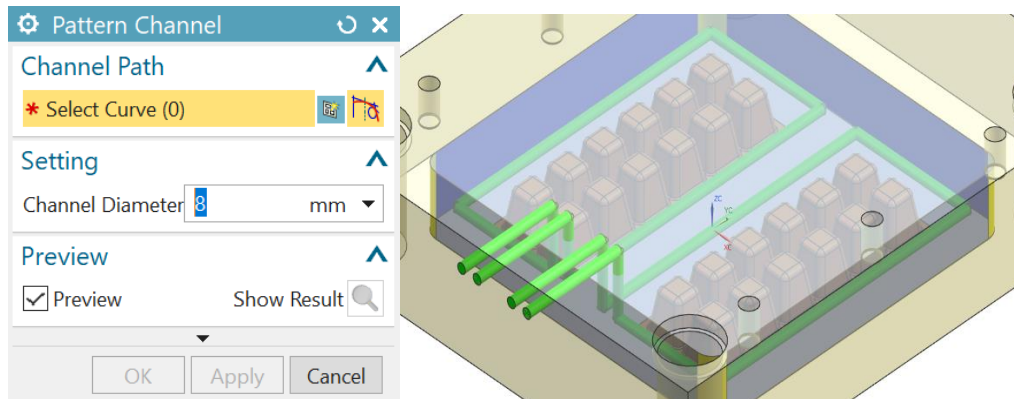


Figure 4.15: Pattern channel design

Step 2: After that, the channel needed to be extended because the cooling channel needed a hole to insert the fluid and also to make the fluid travel inside and outside. Select **Extend Channel**, then select the channel that needs to be extended. After that, choose the extended limit either by distance or by the part's boundary, and click **Apply** to confirm the chosen design.

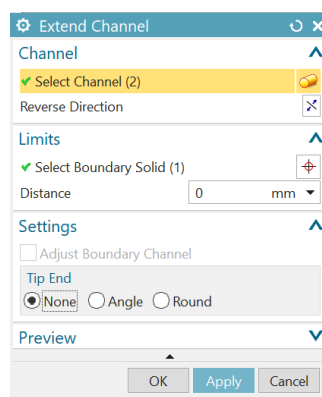


Figure 4.16: Extend Channel tab

Step 3: Setting the cooling circuit is to tell NX which one is the inlet and which one is the outlet. Select **Cooling Circuit** and select a section of the channel as the desired inlet. Then click on the arrow to define the flow of the fluid inside the channel.

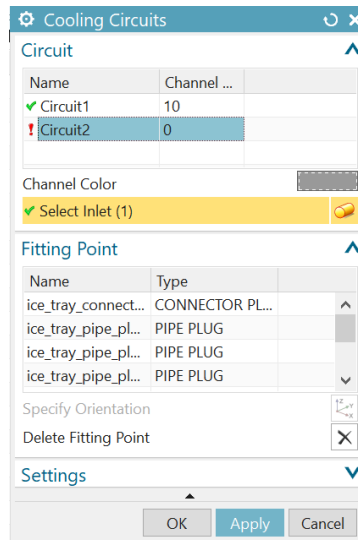


Figure 4.17: Cooling Circuits setting

Step 4: During the design of the cooling channel, several channels cross the interface of the "A" plate and the cavity. This can cause coolant to flow through the gap, so the O-rings should be installed in these locations as a sealing device. To do that, select **Cooling Fitting** and select the channels that are needed to install the O-ring, then click **OK**.

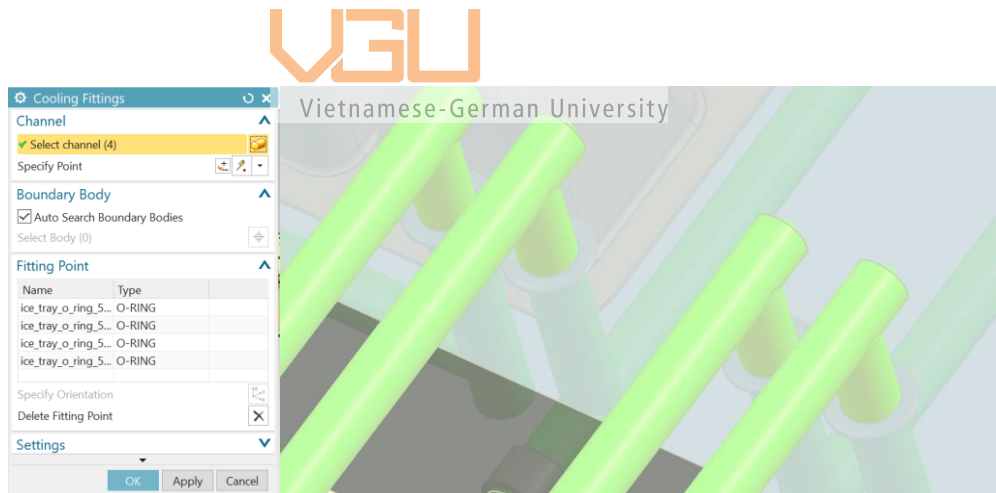


Figure 4.18: O-rings insert

Step 5: To generate a 3D model for all of the symbols of pipe plugs, connectors, and O-rings, go to **Concept Design**, select all, and then click OK. NX will automatically create all of the cooling channel's accessories.

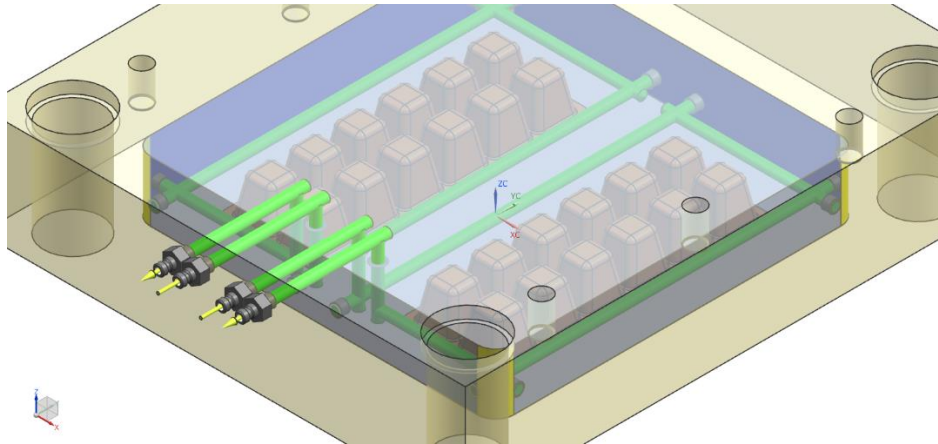


Figure 4.19: Final design of cooling channels

#### 4.6. Ejector system

The following part will demonstrate how to insert ejector pins for this assembly.

Step 1: Select the **Design Ejector Pins** option on the Mold Wizard menu. The base ejector pin design chosen in this thesis is the FUTABA\_MM straight ejector pin.

Step 2: Choose the size of the pin in the catalog. Note that the diameter of the pins should be neither too large, which might leave pin marks on the product, nor too small, which might cause punctures on the surface of the product. Here, the pin's diameter is chosen to be 5 mm and the length to be about 300 mm.

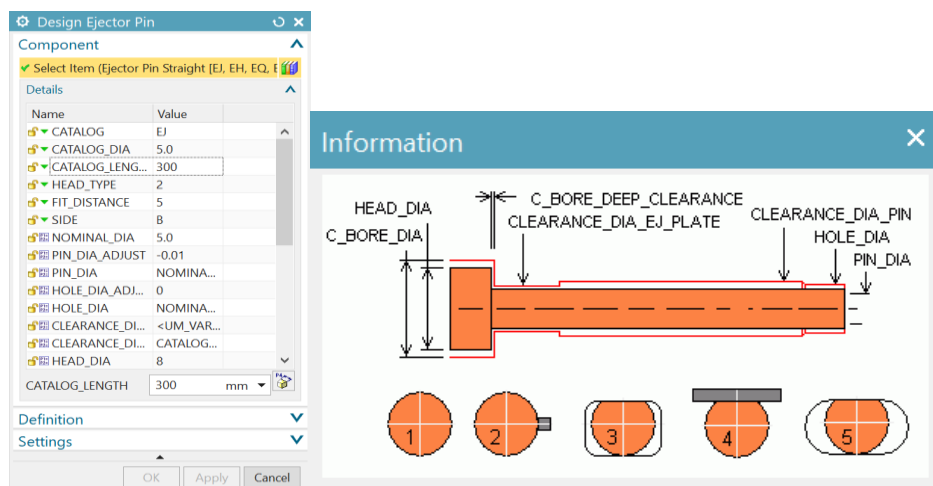


Figure 4.20: Ejector pin design

Step 3: Now for the pin's arrangement on the product, in the **Definition** section of the **Design Ejector Pin** tab, select **Specific Point** and start laying out the placement of the pins. Ejector pins should be applied in places where aesthetics are not required. The

number of pins should not be too large, as this will increase the cost of the mold, but it also should not be too small, as this will cause the pushing force to be unevenly distributed, resulting in defects. After that, click **OK**.

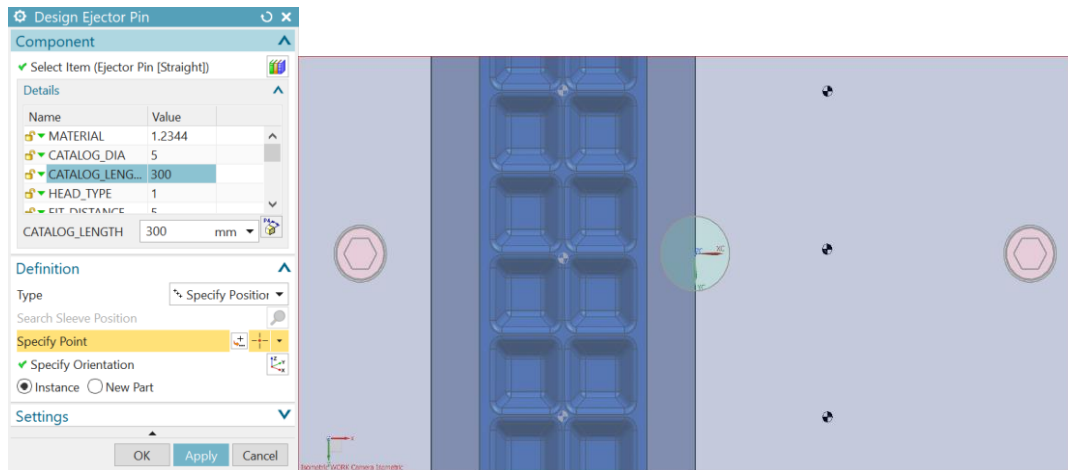


Figure 4.21: Design layout of ejector pins

Until now, the 3D geometry for the ejector pin has not yet been generated; only the symbol has. In order to create this 3D model, go to **Concept Design** and select all pins designed for NX to automatically create a 3D model for the pins.

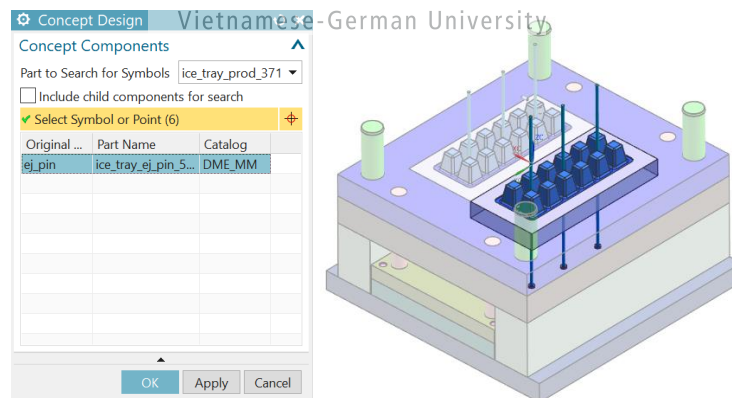


Figure 4.22: Creating the Ejector pins

Step 4: As can be seen, the pin's length is too long, so it needs to be trimmed using **Ejector Pin Post Processing**. In this tab, select all the pins, and then the tool for the trim is the trim surface called **CORE\_TRIM\_SURFACE** (blue color part). Click **OK**, and now the length is automatically trimmed.

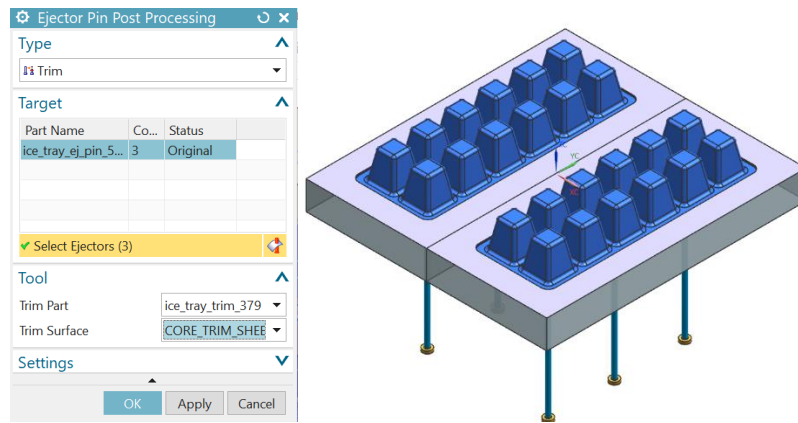


Figure 4.23: Trimmed ejector pins

#### 4.7. Venting system

Last but not least is calculating the design of the vent channel when the product is a PP plastic ice tray. Here, the selected type of venting was air vents through the parting surface. This type of vent is quite easy to design and manufacture.

Parameters needed to design the venting channel:

- Depth of vents:  $d$
- Width of the vent:  $W$
- Vent length:  $L$
- Depth of drain  $D = 20d$

Because the product is made of PP, from Table 2.2, the air vent depth is between 0.0127 and 0.03 mm, so the average depth value of  $d = 0.02$  mm was chosen. From that assumption, the depth of the drain is  $D = 20d = 0.4$  mm, while the parameters  $W$  and  $L$  are taken according to the actual size range, with  $L = 2$  mm and  $W = 4$  mm, respectively.

As stated in Section 2.5, the number of air vents depends on the circumference of the part. The total circumference of the air outlet must be close to 30% of the parting line of the part to ensure the ability to escape. Hence the calculation:

- Circumference of 1 air outlet:  $a = 2*(W + d) = 2*(4+0.02) = 8.04$  mm.
- The approximate length of the parting line of the product:  $A = (225+81) * 2 = 612$  mm.
- Number of air vents:  $X = 0.3A/a = 0.3*(612/8.04) = 22.8 \rightarrow$  Choose  $X = 23$ .



Finally, using the **Extrude** command, the venting layout was sketched and constructed.

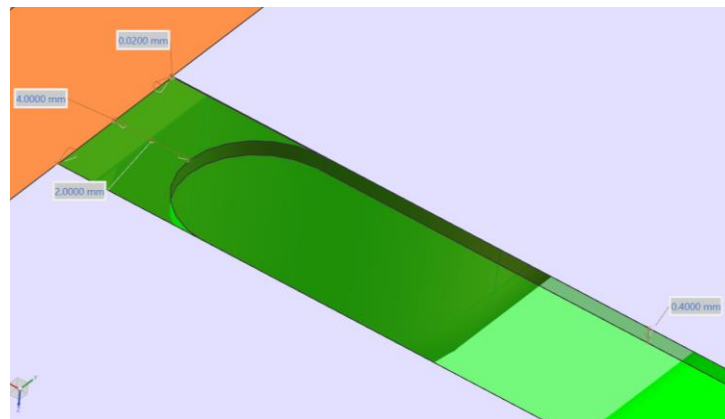


Figure 4.24: Venting channel parameters

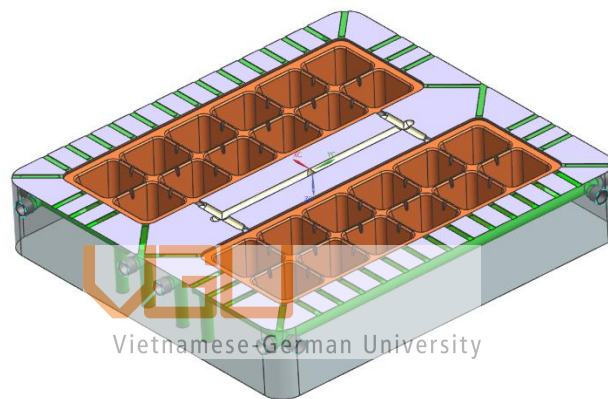


Figure 4.25: Venting system layout

#### 4.8. Final product

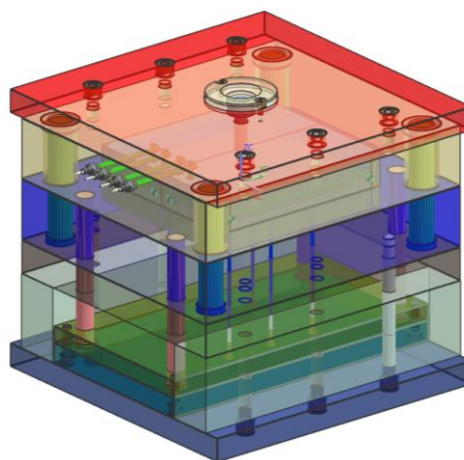


Figure 4.26: Final product

## 5. CONCLUSION

This thesis has studied product design for an injection molding machine, and a simulation of a plastic Ice Tray product design was performed utilizing the program NX 11, as well as the basic theory of injection molding mold design. Moreover, from the start, fundamental plastic material analysis was discussed. After multiple trials, the product design and simulation results for the present capacity of the NX 11 program were attained.

Many assumptions were made regarding the melt simulation and analytical parameters of the product. Values for the calculation of clamp force, estimated injection pressure, and volume flow rate were discovered based on the value ranges of an average injection molding machine. The simulation study is performed on all elements of the melt after examining and identifying the assumptions and parameters of the plastic Ice Tray. As a result, the flow of the melt material, chosen to be polypropylene, is evaluated, and multiple experiments are carried out until an optimal analysis of the melt fill time, clamping force, average velocity, and fill pressure results is produced. The results show a clamp force of 59.86 tons and an operating pressure of 54.529 MPa, both of which are practically possible and feasible for the manufacture of the Plastic Ice Tray utilizing an average load injection molding machine.

Finally, the mold can be created in a proper industrial setting using the best steel material. Mold manufacturing milling machining such as CNC can be used to construct the core and cavity, making the production of the plastic Ice Tray more realistic in injection molding industries.

The project of this thesis could be improved in numerous directions in order to improve the completion of the project, to build a better, more complete "Ice Tray," or to analyze the comparison of mold processing methods. Some suggestions for future improvement of the topic include using multi-plate mold design to improve production efficiency, conducting more research on material properties as well as parameters on injection molding machines to ensure the product has good hardness, high aesthetics, and a variety of colors.

## BIBLIOGRAPHY

- Amon, J. (2008). *Injection Molding: Mold Construction and Part Design*. Retrieved October 20, 2022, from free FlipHTML5 flip book maker:  
<https://fliphtml5.com/tyfe/xhfg/basic>
- Designing Guides: Engineering Plastics*. (2005). Retrieved October 20, 2022, from DSM Engineering Plastics.:  
[http://www.dsm.com/en\\_US/downloads/dep/designbroch05USweb.pdf](http://www.dsm.com/en_US/downloads/dep/designbroch05USweb.pdf)
- Harper, C. A. (2000). *Modern Plastics Handbook*. Technology Seminars, Inc.
- Kazme, D. (2016). *Injection Mold Design Engineering* (2nd ed.). Hanser Publications.
- Leecher, L. (2020, October 2). *Injection Molding Basics: Hot Runner Systems*. Retrieved October 20, 2022, from Echo Engineering - Innovation by Design:  
<https://www.echosupply.com/blog/injection-molding-basics-hot-runner-systems/>
- Menges, G., Michaeli, W., & Mohren, P. (2001). *How to Make Injection Molds* (3rd ed.). Munich, Germany: Hanser Publications.
- Park, H. S., & Dang, X. P. (2012). *Design and simulation-based optimization of cooling channels for plastic injection mold*. C. Volosencu.
- Rosato. (2000). *Injection molding handbook*. Kluwer Academic Publishers.
- Wang, M., Chang, R., & Hsu, C. D. (2018). *Molding Simulation, Theory and Practice*. Hanser Publication.
- Williams, C. (2018). *The 10 Most Popular Injection Molding Materials*. Retrieved October 21, 2022, from Star Rapid: <https://www.starrapid.com/blog/the-ten-most-popular-plastic-injection-molding-materials/>
- Xie, L., Shen, L., & Jiang, B. (2011). *Modelling and Simulation for Micro Injection Molding Process*. Retrieved October 19, 2022, from Research gate:  
[https://www.researchgate.net/figure/1-Processing-cycle-of-conventional-injection-molding-process-Source-Veltkamp\\_fig1\\_221913235](https://www.researchgate.net/figure/1-Processing-cycle-of-conventional-injection-molding-process-Source-Veltkamp_fig1_221913235)