

**Experimental and Numerical Investigation of Key
Parameters in Jet Impingement Condensation for
Optimized Water Production Efficiency**

BY

Turki Shaya Al Ati

A Thesis Presented to the
COLLEGE OF GRADUATE AND INTERDISCIPLINARY STUDIES
KING FAHD UNIVERSITY OF PETROLEUM & MINERALS
DHAHRAN, SAUDI ARABIA

In Partial Fulfillment of the
Requirements for the Degree of

MASTER OF SCIENCE

In

MECHANICAL ENGINEERING

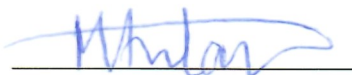
December 2024

KING FAHD UNIVERSITY OF PETROLEUM & MINERALS
DHAHRAN- 31261, SAUDI ARABIA
COLLEGE OF GRADUATE AND INTERDISCIPLINARY STUDIES

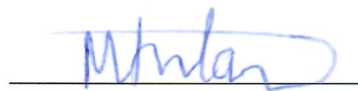
This thesis, written by **TURKI SHAYA AL ATI** under the direction of his thesis advisor and approved by his thesis committee, has been presented and accepted by the Dean, College of Graduate and Interdisciplinary Studies, in partial fulfillment of the requirements for the degree of **MASTER OF SCIENCE IN MECHANICAL ENGINEERING**.



Dr. Ali Alshehri
(Advisor)



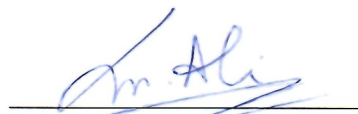
Dr. Mohamed Antar
Department Chairman



Dr. Mohamed Antar
(Member)



Dr. Suliman S. Al-Homidan
Dean, College of Graduate and
Interdisciplinary Studies



Dr. Hafiz Ali
(Member)

Date



© Turki Shaya Al Ati

2024

Dedication

To my father, **Eng. Shaya Al Ati**, whose inspiration and guidance have been a constant source of motivation throughout my journey; to my mother, **Guzail Al Ati**, whose unwavering love and heartfelt prayers have always been my strength and support in achieving this milestone; to my brothers and sister **Abdullah, Eylaf, Zyad, and Mishal** for their encouragement and belief in me; to my beloved wife, **Sadeem**, whose endless support, patience, and encouragement have made it possible for me to focus and persevere in this journey; and to my children **Shaya, Melad, and Shamekh**, the future scientists and leaders, whose bright smiles and dreams inspire me to strive for excellence. This work is dedicated to all of you, with love and gratitude for your faith in me and your role in making this achievement possible.

ACKNOWLEDGMENTS

I would like to express my deepest gratitude to my advisor, **Dr. Ali Alshehri**, for his invaluable guidance, insightful advice, and unwavering availability throughout the course of my research. His mentorship has been a cornerstone of my academic journey, providing the support and encouragement needed to achieve this milestone.

I am also profoundly thankful to my committee members, **Prof. Mohamed Anter** and **Dr. Hafiz Ali**, for their valuable feedback and continued support. Their expertise and constructive suggestions have been instrumental in shaping the direction of my research and ensuring its success. Additionally, I extend my sincere appreciation to **Mr. Saad Alzahrani**, the lab engineer, for his steadfast support and availability, which greatly contributed to the smooth execution of my experimental work.

Special thanks to my brother, **Bander ALOsaimi**, who shared this journey with me from the very beginning, starting with our admission and culminating in this achievement. His companionship and encouragement have been a source of strength and motivation throughout this incredible experience.

TABLE OF CONTENTS

ACKNOWLEDGMENTS	V
TABLE OF CONTENTS	VI
LIST OF TABLES	IX
LIST OF FIGURES	X
LIST OF ABBREVIATIONS	XII
ABSTRACT.....	XVI
ملخص الرسالة.....	XVIII
CHAPTER 1 INTRODUCTION.....	1
1.1 Evaporation	2
1.1.1. Natural Evaporation.....	3
1.1.2. Forced or Enhanced Evaporation	3
1.1.3. Flash Evaporation	4
1.1.4. Multiple Effect Evaporation (MEE)	4
1.1.5. Bubble Column Evaporation.....	4
1.2. Condensation	5
1.2.1. Homogeneous Condensation.....	6
1.2.2. Heterogeneous Condensation	7
1.3. Condenser	8
1.3.1 Condensers Cooled by Water	9
1.3.2 Condensers Cooled by Air	9
1.3.3 Evaporative Condensers	9

1.4.	Jet Impingement Technique	9
1.5.	RESEARCH MOTIVATION	10
1.6.	RESEARCH OBJECTIVES	11
1.7.	THESIS ORGANIZATION	12
CHAPTER 2 LITERATURE REVIEW		13
2.1.	INTRODUCTION.....	13
2.2.	Jet impingement condenser	13
2.3.	Physics of Jet Impingement Condensation	14
2.4.	Influence of Geometrical Variables and other parameters on condensation efficiency	18
2.4.1.	Jet to Plate Spacing (H/D):.....	19
2.4.2.	Nozzle to nozzle Spacing (S/D):	22
2.4.3.	Jet Pattern (Inline vs. Staggered):.....	25
2.5.	Effect of Area of the Target Surface:	29
2.6.	Effect of Temperatures.....	32
2.7.	Flow Dynamics	35
2.8.	Two-Phase Bubble Columns	37
CHAPTER 3 METHODOLOGY		39
3.1.	INTRODUCTION.....	39
3.2.	EXPERIMENTAL METHODOLOGY	39
3.2.1.	UNCERTAINTY ANALYSIS	45
3.3.	NUMERICAL METHODOLOGY.....	46
3.3.1.	Problem Statement.....	46
3.3.2.	Boundary Conditions	47
3.3.3.	Governing Equations for CFD Simulation.....	49
3.3.4.	Computational power.....	53

3.3.5. Mesh Sensitivity Analysis	54
3.3.6. Thermophysical Properties of Working Fluids	55
3.3.7. Simulation Setting	55
3.3.8. Validation of the model.	56
CHAPTER 4 RESULTS AND DISCUSSION.....	57
4.1. EXPERIMENTAL RESULTS AND DISCUSSION	57
4.1.1. Bubble Column Analysis	57
4.1.2. Nusselt Number Analysis	58
4.1.3. Sherwood Number Analysis	61
4.1.4. Comparative Analysis of Configurations	63
4.2. NUMERICAL RESULTS AND DISCUSSION	64
4.2.1. Nusselt Number Analysis	64
4.2.2. Sherwood Number Analysis	81
CHAPTER 5 CONCLUSION	86
REFERENCES.....	87
VITAE	95

LIST OF TABLES

Table 1 Review of single and multiple jet impingement systems.....	16
Table 2 Review of nozzle to plate spacing.....	21
Table 3 Review of nozzle to nozzle spacing	24
Table 4 Review of Jet Pattern	27
Table 5 Reviews of the impact of the area of the target surface.....	30
Table 6 Reviews of the impact of the temperature	33
Table 7 Summary of the experimental configurations	43
Table 8 The uncertainty approximation values	46
Table 9 Properties of fluids.	55

LIST OF FIGURES

Figure 1	Concept of evaporation and condensation.....	2
Figure 2	Classification of evaporation techniques.....	3
Figure 3	Classification of condensation (reconstructed from.....	6
Figure 4	Jet impingement geometric parameters.....	20
Figure 5	Jets Pattern (a) inline, (b) staggered.....	27
Figure 6	Two phases bubble column and JIC	40
Figure 7	Jet impingement condenser model consists of 1) Air accumulation area, 2) Gasket, 3) Perforated plate, 4) Spacer, 5) Spacer 6) Copper plate, 7) Gasket, 8) Chilled water channel.	40
Figure 8	Experimental Setup.....	41
Figure 9	Computational domain.	47
Figure 10	Boundary condition.....	48
Figure 11	Mesh generation on computational domain with different element sizes (a) 0.001, (b) 0.0005, (c) 0.00025 and (d) 0.00015.....	54
Figure 12	Model validation.....	56
Figure 13	The relationship between the heater temperatures and T_{dry}	58
Figure 14	(a-c) Nu vs Ja of all configurations respectively.....	60
Figure 15	(a-c) Sh vs Ja of all configurations respectively.	63
Figure 16	Nusselt Number Vs Temperature.	65
Figure 17	Contours of (a) H_2O Mass Fraction and (b) Temperature Distribution on Plane for Jet Impingement Flow at varying temperature test of inlet vapor. Streamline Visualization of (c) H_2O and (d) Temperature Mass Fraction Distribution in Jet Impingement Configuration at various temperature of inlet vapor	66
Figure 18	Nusselt Number Vs Surface Area.	67
Figure 19	Contours of (a) H_2O Mass Fraction and (b) Temperature Distribution on Plane for Jet Impingement Flow at area $35*35mm^2$. Streamline Visualization of (c) H_2O and (d) Temperature Mass Fraction Distribution in Jet Impingement Configuration at area $35*35mm^2$	68
Figure 20	Contours of (a) H_2O Mass Fraction and (b) Temperature Distribution on Plane for Jet Impingement Flow at area $75*75mm^2$. Streamline Visualization of (c) H_2O and (d) Temperature Mass Fraction Distribution in Jet Impingement Configuration at area $75*75mm^2$	69
Figure 21	Nusselt Number Vs S/D ratio.	70
Figure 22	Contours of (a) H_2O Mass Fraction and (b) Temperature Distribution on Plane for Jet Impingement Flow at $S/D=2$. Streamline Visualization of (c) H_2O and (d) Temperature Mass Fraction Distribution in Jet Impingement Configuration at $S/D=2$	71

Figure 23	Contours of (a) H₂O Mass Fraction and (b) Temperature Distribution on Plane for Jet Impingement Flow at S/D=15. Streamline Visualization of (c) H₂O and (d) Temperature Mass Fraction Distribution in Jet Impingement Configuration at S/D=15.	72
Figure 24	Contours of (a) H₂O Mass Fraction and (b) Temperature Distribution on Plane for Jet Impingement Flow at S/D=18. Streamline Visualization of (c) H₂O and (d) Temperature Mass Fraction Distribution in Jet Impingement Configuration at S/D=18.	73
Figure 25	Nusselt Number Vs H/D.....	74
Figure 26	Contours of (a) H₂O Mass Fraction and (b) Temperature Distribution on Plane for Jet Impingement Flow at H/D=2. Streamline Visualization of (c) H₂O and (d) Temperature Mass Fraction Distribution in Jet Impingement Configuration at H/D=2.....	75
Figure 27	Contours of (a) H₂O Mass Fraction and (b) Temperature Distribution on Plane for Jet Impingement Flow at H/D=20. Streamline Visualization of (c) H₂O and (d) Temperature Mass Fraction Distribution in Jet Impingement Configuration at H/D=20.....	76
Figure 28	Nusselt Number Vs Velocity(m/s).	77
Figure 29	Contours of (a) H₂O Mass Fraction and (b) Temperature Distribution on Plane for Jet Impingement Flow at v = 5 m/s. Streamline Visualization of (c) H₂O and (d) Temperature Mass Fraction Distribution in Jet Impingement Configuration at v = 5 m/s.....	78
Figure 30	Contours of (a) H₂O Mass Fraction and (b) Temperature Distribution on Plane for Jet Impingement Flow at v = 45 m/s. Streamline Visualization of (c) H₂O and (d) Temperature Mass Fraction Distribution in Jet Impingement Configuration at v = 45 m/s.....	79
Figure 31	Contours of (a) H₂O Mass Fraction and (b) Temperature Distribution on Plane for Jet Impingement Flow at v = 90 m/s. Streamline Visualization of (c) H₂O and (d) Temperature Mass Fraction Distribution in Jet Impingement Configuration at v = 90 m/s.....	80
Figure 32	Sherwood Number Vs Temperature.	81
Figure 33	Sherwood Number Vs Area.	82
Figure 34	Sherwood Number Vs S/D ratio.	83
Figure 35	Sherwood Number Vs H/D ratio.....	84
Figure 36	Sherwood Number Vs Velocity (m/s).	84

LIST OF ABBREVIATIONS

Abbreviations

JIC	:	Jet Impingement Condenser
DI	:	Deionized
MSF	:	Multi Stag Flas
MEE	:	Multi Effect Evaporation
HDH	:	Humidification Dehumidification
CD	:	Convergent Divergent
CFM	:	Cubic Feet Per Minute
HDPE	:	High Density Polyethylene
H/D	:	Ratio of the nozzle-to-surface distance (H) to the diameter of the nozzle (D)
S/D	:	Ratio of the spacing between adjacent nozzles (S) to the diameter of the nozzle (D)
CFD	:	Computational Fluid Dynamics
CPU	:	Central Processing Unit
UHD	:	Ultra-High Definition
ECC	:	Error Correcting Code
RAM	:	Random Access Memory
GPU	:	Graphic Processing Unit
3D	:	Three Dimensional

Dimensionless Numbers

Nu	:	Nusselt Number	Re	:	Nusselt Number
Sh	:	Sherwood Number	Ja	:	Jakob number

Fundamental Thermo Fluid Properties

$\dot{m}''$	: Mass flux
q''	: Heat transfer per unit area [W/m ²]
$\dot{m}$	: Mass transfer rate [Kg/s]
F_{VOL}	: Volume Force [N]
T_{∞}	: Temperature of Incoming Humid air [K]
T_s	: Saturation Temperature [K]
C_{pl}	: Specific heat of Liquid [kj/kgK]
C_{pv}	: Specific Heat of Vapor [kj/kgK]
ω_{∞}	: Vapor mass fraction of incoming humid air
ω_s	: Vapor mass fraction at the condenser surface
ω_i	: Vapor mass fraction at a given condition
$\nabla \cdot \mathbf{v}$	: Divergence of the velocity field (used in continuity equation)
Y_i	: Mass fraction of species i
ω_i	: Rate of generation or consumption of species i
D_{aw}	: Binary mass diffusivity of water vapor into air
D_{crit}	: Critical Diameter [mm]
h_{fg}	: Latent Heat [kj/kgK]
μ_l	: Viscosity of Liquid [Ns/m ²]
μ_v	: Viscosity of vapor [Ns/m ²]
σ_l	: Surface tension of liquid [N/m]
ρ_l	: Density of Liquid [kg/m ³]
ρ_v	: Density of vapor [kg/m ³]
k_l	: Conductivity of liquid [W/mk]
k_v	: Conductivity of vapor [W/mk]
C_{Pl}	: Specific Heat of Liquid [kj/kgK]
v_i	: Initial Specific Volume [m ³ /kg]

v_f	: Final Specific Volume [m ³ /kg]
g	: Gravity [m/s ²]
T	: Temperature [K]
E	: Energy [J]
t	: Time [s]
D	: Diameter [mm]
P	: Pressure [Pa]
V	: Volume (m ³)
A	: Area (m ²)
∇T	: Gradient Temperature

Derivative Terms

$\frac{\partial v^{\rightarrow}}{\partial t}$	: Change of Velocity per unit time
$\frac{\partial T}{\partial t}$	: Change of Temperature per unit time
$\frac{\partial(\rho Y_i)}{\partial t}$	: Change of density and species mass fraction per unit time
$\frac{\Delta m}{\Delta t}$	: Change of mass per unit time

Greek Symbol

σ	: Surface Tension
τ	: Shear Stress
α	: Void Fraction
μ	: Viscosity

Subscript

l	:	Liquid
v	:	Vapor
s	:	Saturation
crit	:	Critical
dry	:	Dry
cond	:	Condensation
sat	:	Saturated

ABSTRACT

Full Name : Turki Shaya Al Ati

Thesis Title : EXPERIMENTAL AND NUMERICAL INVESTIGATION OF KEY PARAMETERS IN JET IMPINGEMENT CONDENSATION FOR OPTIMIZED WATER PRODUCTION EFFICIENCY

Major Field : Mechanical Engineering

Date of Degree : December 2024

This study presents an experimental and numerical investigation of a jet impingement condenser (JIC) aimed at evaluating the effects of critical design parameters on heat and mass transfer efficiency. The experimental work found that Configuration 1, which is characterized by a jet-to-plate distance of 2 mm, a 35 mm x 35 mm target plate area, and a nozzle spacing of 7 mm, demonstrated the highest performance among the tested configurations. This configuration achieved the greatest Nusselt (Nu) and Sherwood (Sh) numbers, attributed to the concentrated and effective jet impingement, which enhanced heat and mass transfer. Complementing the experimental efforts, numerical simulations using ANSYS were employed to explore a broader range of parameters, including variations in jet-to-plate distance (H/D), nozzle spacing (S/D), target plate area, and fluid velocity. The numerical results provided detailed insights and identified the best possible values, such as an H/D ratio of 2 and an S/D ratio of 15, which significantly enhanced both heat and mass transfer efficiencies in the system. These findings highlight the importance of systematically varying parameters to achieve the best performance.

This comprehensive investigation emphasizes the need for parameter optimization when designing compact, high-performance condensers, and offers valuable insights for

industrial applications requiring efficient thermal management and sustainable water production systems.

ملخص الرسالة

الاسم الكامل: تركي بن شايح ال عاتي

عنوان الرسالة: دراسة تجريبية وعددية للمتغيرات الرئيسية في تكاثف النفط لتحقيق كفاءة مثلى لإنتاج المياه

التخصص: الهندسة الميكانيكية

تاريخ الدرجة العلمية: ديسمبر 2024

تتناول هذه الدراسة تحقيقًا تجريبيًا وعدديًا لمكثف يعتمد على تقنية الارتطام النفث (JIC) لتقييم تأثير المتغيرات التصميمية الرئيسية على كفاءة انتقال الحرارة والكتلة. وقد أظهرت النتائج التجريبية أن التكوين الأول، الذي يتضمن مسافة بين النفثات واللوح (H/D) مقدارها 2 مم، ومساحة لوح مستهدفة تبلغ 35×35 مم، وتباعد بين الفوهات (S/D) بمقدار 7 مم، هو الأكثر كفاءة بين التكوينات التي تم اختبارها. وقد حقق هذا التكوين أعلى القيم لعدد نوسلت (Nu) وعدد شيرودود (Sh)، ويُعزى ذلك إلى تأثير الارتطام النفث المركز والفعال، الذي ساهم في تعزيز انتقال الحرارة والكتلة.

إلى جانب العمل التجريبي، تم استخدام محاكاة عددية بواسطة برنامج ANSYS لاستكشاف نطاق أوسع من المتغيرات، بما في ذلك مسافة النفثات إلى اللوح (H/D)، وتباعد الفوهات (S/D)، ومساحة اللوح، وسرعة التدفق. وقد أظهرت النتائج العددية قيمًا مثلى، مثل نسبة H/D مقدارها 2 ونسبة S/D مقدارها 15، التي عززت بشكل كبير كفاءة انتقال الحرارة والكتلة في النظام. تؤكد هذه النتائج أهمية ضبط المتغيرات بعناية لتحقيق الأداء الأمثل.

تُبرز هذه الدراسة الشاملة أهمية تحسين المتغيرات عند تصميم مكثفات مدمجة وذات أداء عالٍ، وتوفر رؤى قيمة للتطبيقات الصناعية التي تتطلب إدارة حرارية فعالة وأنظمة إنتاج مياه مستدامة.

CHAPTER 1

INTRODUCTION

Water is an essential resource that sustains all forms of life and supports critical industrial, agricultural, and environmental processes. As global water demands continue to rise due to population growth, industrialization, and changing climate patterns, the importance of optimizing water production, management, and recycling has become paramount[1]. Since many people worldwide are already experiencing a water shortage, technological advancements in water production methods have emerged as critical solutions to mitigate this crisis[2]. Among these methods, evaporation and condensation processes play a major role in producing freshwater from saline or polluted sources, supporting desalination, wastewater treatment, and sustainable water management initiatives[3]. These fundamental thermodynamic processes can be coupled and optimized through innovative designs and techniques, leading to more efficient and accessible water production systems. Figure 1 shows conceptual frameworks for evaporation and condensation processes.

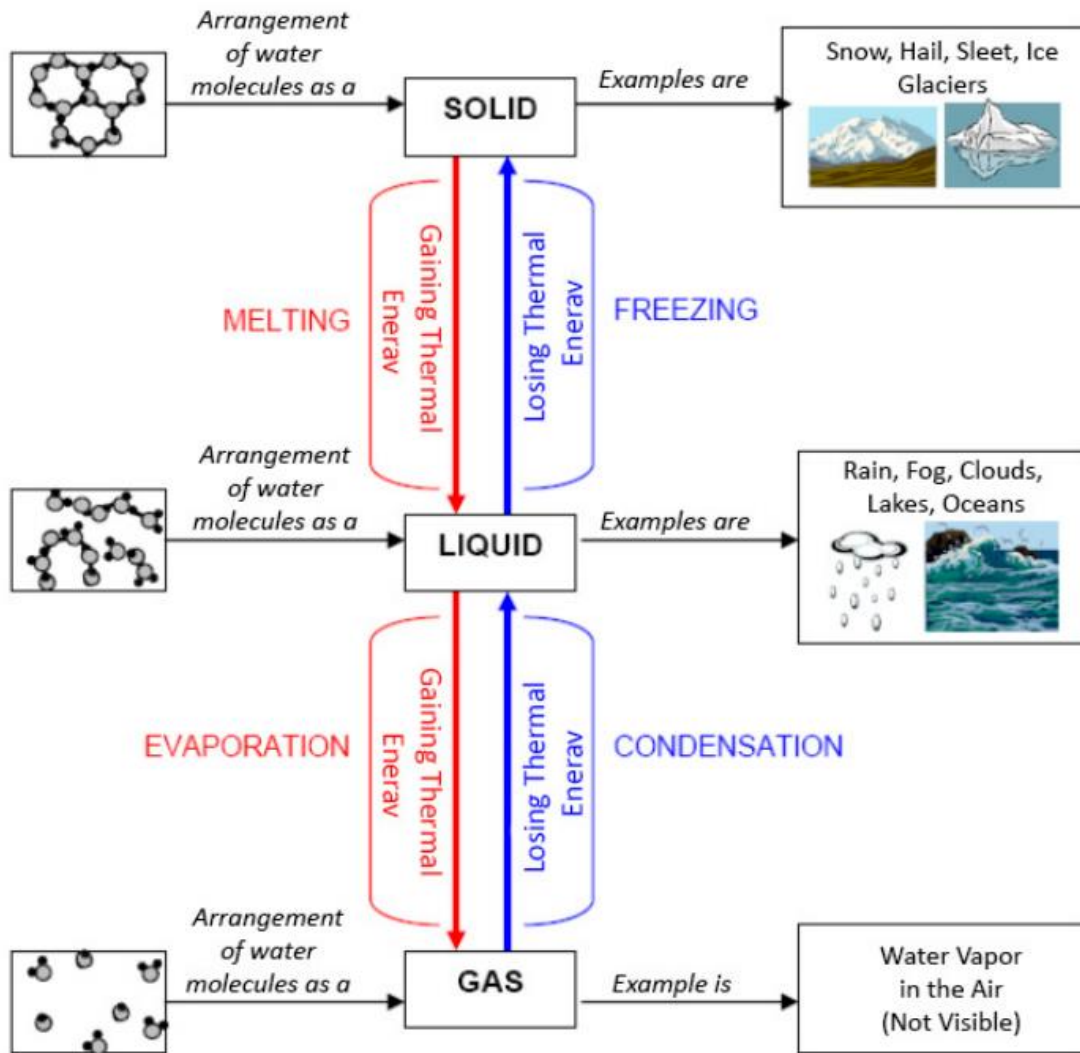


Figure 1 Concept of evaporation and condensation[4]

1.1 Evaporation

Evaporation is the process of converting a liquid into vapor by supplying energy, is a vital natural mechanism that drives the water cycle. This phase change occurs when molecules at the liquid surface gain sufficient kinetic energy to overcome intermolecular forces, transitioning into the gaseous phase[4], [5]. In nature, evaporation plays a key role in transferring moisture from oceans, lakes, and rivers into the atmosphere, forming the basis of precipitation and maintaining global hydrological balance[5]. However, in industrial and

engineered settings, evaporation is manipulated and controlled to achieve various objectives, including concentration of solutions, separation of components, and desalination[6]. There are several types of evaporation techniques[7], [8], [9], each suited to specific applications and conditions following are some of evaporation techniques:

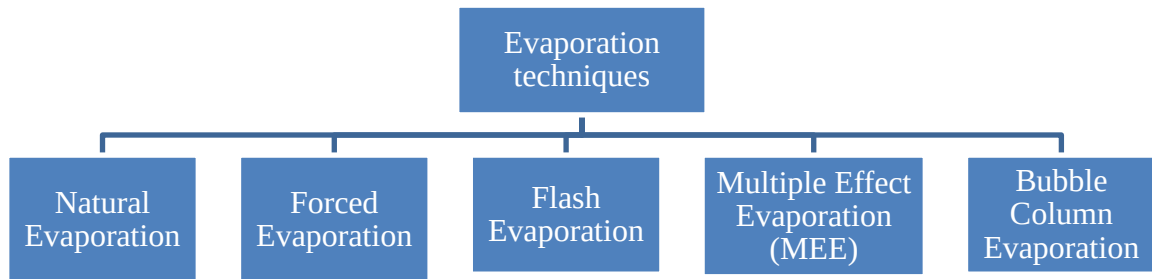


Figure 2 Classification of evaporation techniques[7], [8], [9]

1.1.1. Natural Evaporation

This type of evaporation occurs passively under ambient conditions, driven only by solar energy and environmental factors. It is common in salt production from seawater and drying processes. However, it is slow and weather-dependent, limiting its industrial viability[10].

1.1.2. Forced or Enhanced Evaporation

In contrast to natural evaporation, forced evaporation involves applying external energy, usually in the form of heat, to accelerate the phase change. This method is widely used in industrial applications such as chemical processing, waste treatment, and desalination

plants. Evaporators in this category often utilize heating elements, mechanical vapor recompression, or heat exchangers to achieve efficient evaporation[11].

1.1.3. Flash Evaporation

This process involves subjecting a liquid to a sudden reduction in pressure, causing it to evaporate rapidly. Flash evaporation is used extensively in desalination systems, particularly in multi-stage flash (MSF) desalination, where seawater undergoes sequential flash evaporation in low-pressure stages to yield distilled water[12].

1.1.4. Multiple Effect Evaporation (MEE)

In this method, vapor generated from one evaporative stage is used as the heating source for the next stage, thereby conserving energy. MEE is a common approach in large-scale industrial operations, including desalination and concentration processes, due to its efficiency in reducing energy consumption[13].

1.1.5. Bubble Column Evaporation

An innovative and specialized technique, bubble column evaporation involves bubbling gas (typically air or steam) through a liquid column, enhancing mass and heat transfer. The turbulence created by the rising bubbles increases the surface area for evaporation, making this technique particularly effective for concentrated or viscous liquids. Recent research has explored the use of dual bubble columns to optimize evaporation rates, with significant implications for water treatment and chemical engineering applications[14].

The concept of using two bubble columns for evaporation is a notable advancement aimed at maximizing the efficiency and output of the evaporation process. In such a configuration, two separate columns operate in cycle, allowing for greater control over variables such as

gas flow rate, temperature, and liquid composition. The primary advantage lies in the increased interfacial area between the liquid and gas phases, as well as the enhanced mixing and turbulence introduced by the bubbling action. This leads to faster evaporation rates and more effective utilization of energy[8].

The first column typically acts as a pre-concentration stage, where initial evaporation occurs under controlled conditions. The vapor then passes into the second column, where further evaporation is achieved, often under different conditions tailored to the specific application. This staged approach allows for precise optimization of each stage, improving overall system efficiency. Additionally, the ability to manipulate variables independently in each column provides flexibility in handling varying feed compositions and operational requirements[8].

Dual bubble column systems have found applications in desalination, where they offer a scalable and energy-efficient solution for converting saline water into freshwater. They are also used in chemical industries for concentrating solutions and separating volatile components. The use of bubble columns for evaporation can be integrated with other processes, such as distillation or crystallization, providing a versatile tool for complex separation tasks[15], [16].

1.2. Condensation

Condensation, the inverse process of evaporation, involves the conversion of vapor into liquid by releasing energy. In the natural environment, condensation is responsible for cloud formation, dew, and rainfall. In engineered systems, it plays a critical role in heat exchangers, power plants, refrigeration cycles, and water production technologies. The

efficiency and effectiveness of condensation processes directly impact the performance of various industrial and environmental applications[17]. The different types of condensation phenomena as illustrated in figure 3:

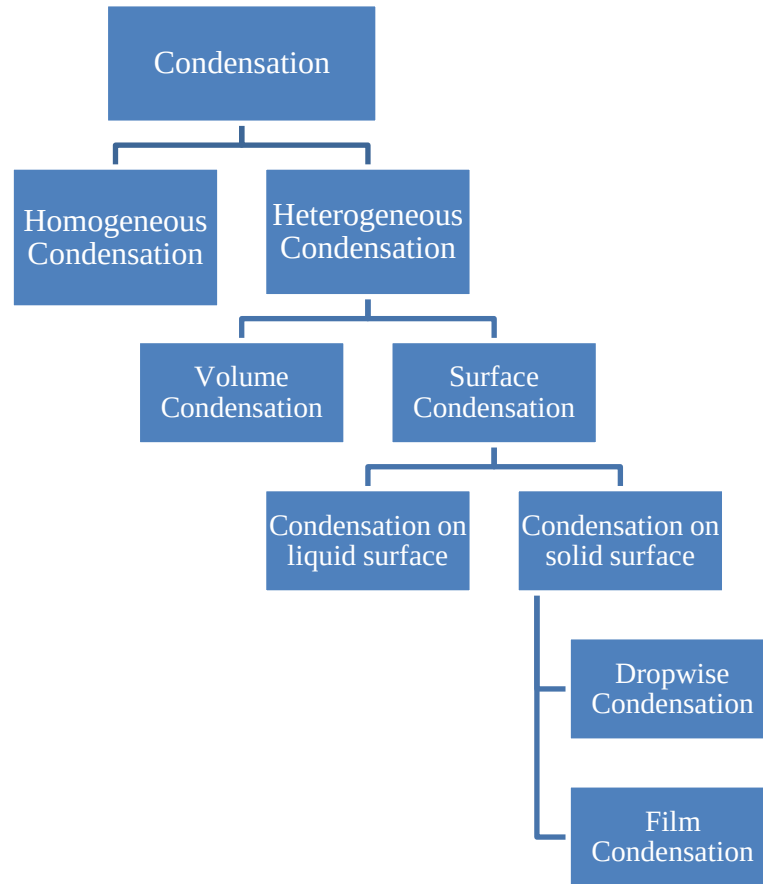


Figure 3 Classification of condensation (reconstructed from [18])

1.2.1. Homogeneous Condensation

Homogeneous condensation occurs in the absence of foreign substances and results purely from the random fluctuations in the motion of vapor molecules. It is a stochastic process that happens without any external nuclei. While it provides theoretical insight, this type of condensation rarely plays a significant role in practical applications due to the specific conditions required for it to occur[18].

1.2.2. Heterogeneous Condensation

Heterogeneous condensation, which is more relevant in practical applications, involves the presence of foreign materials (either liquid or solid) that act as nuclei to facilitate the condensation process. This type is further classified into (A) volume (bulk) condensation and (b) surface condensation[18].

A. Volume (Bulk) Condensation

This type of condensation happens in the vapor space where small particles, such as mist or fog, serve as nuclei for the condensation process. Although this phenomenon is essential for understanding natural processes like cloud formation, it has limited industrial applications. However, it can pose problems in specific systems, such as sodium vapor deposition in liquid-metal-cooled reactors, where it leads to operational challenges[18].

B. Surface Condensation

Surface condensation occurs when vapor condenses on either a liquid or solid surface. Surface condensation is further subdivided based on the type of surface involved. Condensation on liquid surfaces includes processes like direct contact condensation on jets, drops, or films of liquid[18].

B.1. Direct Contact Condensation

In this process, vapor comes into direct contact with a liquid phase, leading to rapid condensation. Direct contact condensers are often used in gas scrubbing, power generation, and desalination, offering advantages in terms of simplicity and efficiency[18], [19].

B.2. Condensation on solid surfaces

Involves vapor condensing onto a solid wall, leading frequently to the surface's liquid layer forming. Within solid surface condensation, two key modes are recognized: dropwise condensation, where the liquid forms discrete droplets, and film condensation, where the liquid spreads out as a continuous film[18].

B.2.1. Filmwise Condensation

In this mode, a thin liquid layer is formed by the condensate on the surface. While filmwise condensation is common in practical applications, it tends to reduce heat transfer efficiency because of the insulating effect of the film. It is typically encountered in heat exchangers and refrigeration systems[17], [18].

B.2.2. Dropwise Condensation

Dropwise condensation occurs when discrete droplets form on the condensing surface, resulting in increased heat transfer rates. This type of condensation is desirable in many applications but requires specially treated surfaces to maintain droplet behavior[17], [18].

1.3. Condenser

A condenser is a cooling device, commonly referred to as a heat rejector, used in refrigeration systems and power plants to condense refrigerant vapor or steam into a subcooled liquid. The process involves three stages: de-superheating, where excess heat is removed; condensation, where the vapor mostly converts to liquid; and subcooling, which stabilizes the liquid to prevent it from reverting to vapor. Condensers are generally categorized into water-cooled, air-cooled, and evaporative types[20], [21].

1.3.1 Condensers Cooled by Water

Water is circulated in water cooled condensers to absorb heat from refrigerant vapor. On the outside of the tubes, the vapor condenses, transferring heat to the water inside, which then releases the heat into the atmosphere. These systems are known for their efficient heat transfer capabilities [22], [23], [24].

1.3.2 Condensers Cooled by Air

Condensers cooled by air use ambient air to cool and condense vapor within finned tubes. Heat is removed as air flows over the tubes, with the process relying on the surrounding air temperature. The key benefit of air-cooled condensers is their independence from cooling water, making them simpler to operate [23], [24].

1.3.3 Evaporative Condensers

Evaporative condensers remove heat by wetting the condenser coil with recirculating water while air blows over it. As some water evaporates, it draws heat from the vapor inside the coil, leading to condensation. This method effectively combines principles from both air- and water-cooled systems[25] .

1.4. Jet Impingement Technique

This technique uses high-speed vapor jets directed onto a cooled surface, causing rapid condensation with high heat transfer rates, which are ideal for compact and efficient condensers[26]. Besides water production, condensation is essential in processes like heat exchange, chemical mixture, and refrigeration, making it critical in industries such as chemical and food processing. Jet impingement condensers, known for their compact

design and high efficiency, are particularly useful in small-scale applications where space is limited, and high performance is required[27].

These condensers are effective in micro-scale desalination, electronics cooling, and systems in resource-limited environments. Their adjustable design allows precise control over condensation by modifying factors like jet velocity, nozzle configuration, temperature and surface properties, optimizing water production[28]. Additionally, combining these condensers with enhanced evaporation methods, like dual bubble columns, offers a promising way to improve condensation rate as it will be conducted in this research.

Jet impingement is a complicated heat transfer process affected by multiple interacting variables. To understand this process, research must focus on analyzing a few key parameters rather than attempting to study all variables simultaneously, which would require excessive experiments[29]. This study's experimental work will target three main parameters: copper sheet area, spacer length, and varying vapor temperature the numerical analysis will expand on these by also considering air flow rate, jet diameter, and pitch to evaluate their combined effect on condensation performance.

The research aims to identify the ideal combination of parameters for maximizing water production efficiency using a small jet impingement condenser (JIC). The study focuses on determining the most effective parameter configuration, with the objective of developing design guidelines to improve the performance of JIC systems.

1.5. RESEARCH MOTIVATION

The motivation for this research lies in identifying the best configuration of jet impingement condensers (JIC) that achieves the highest heat and mass transfer efficiency,

addressing the limited studies focusing on this specific application. By systematically investigating key parameters such as jet-to-plate distance, nozzle spacing, and target surface area, this study aims to fill a significant gap in understanding the interplay of design and operational factors in JIC systems. The goal is to advance knowledge in this underexplored field and contribute to the development of compact, high-performance solutions for sustainable water production.

1.6. RESEARCH OBJECTIVES

1. To design and develop an experimental setup to investigate how key variables affect the condensation rate in a small jet impingement condenser (JIC).
2. To conduct numerical modeling of the jet impingement condensation process using ANSYS. The numerical analysis will complement the experimental investigation by providing deeper insight into the heat transfer and fluid dynamics involved in the condensation process. By comparing experimental data with simulation results, this objective seeks to validate the numerical model and use it to predict JIC system behavior under a wider range of conditions than those tested experimentally.
3. To identify the best combination of parameters for maximizing water production efficiency using JIC. Aiming to determine the most effective configuration. The goal is to develop guidelines or design principles for creating more efficient JIC systems, which could have important applications in industrial processes where effective water condensation is crucial.

1.7. THESIS ORGANIZATION

This research consists of several chapters, each addressing a critical aspect of the study. Chapter 2 presents a comprehensive literature review, highlighting key studies on jet impingement technique, with a focus on identifying gaps in understanding the interplay of parameters such as jet-to-plate distance, nozzle spacing, and surface configurations. Chapter 3 discusses the experimental and numerical methodologies employed, detailing the design of the experimental setup, the parameters tested, and the computational modeling approaches used to simulate and analyze the condensation process. Chapter 4 presents and analyzes the experimental and numerical results, comparing different configurations and identifying trends in heat and mass transfer efficiencies. Finally, the concluding chapter synthesizes the findings to determine the best configurations and parameters, providing practical insights for enhancing jet impingement condensation systems.

CHAPTER 2

LITERATURE REVIEW

2.1. INTRODUCTION

This chapter provides a comprehensive literature review, focusing on key studies related to the jet impingement technique. It highlights the influence of design parameters on heat and mass transfer in various applications.

2.2. Jet impingement condenser

Jet impingement technique is utilized in numerous industrial applications requiring efficient heat transfer, such as in cooling systems, desalination processes, and electronics thermal management systems[27]. This technique leverages high-velocity fluid jets directed onto a target surface to enhance heat and mass transfer, making it especially effective for promoting condensation. Compared to traditional cooling methods, jet impingement offers significant advantages, including the ability to achieve localized high heat flux and better control over temperature distribution[30], [31].

Although these advantages, the optimization of jet impingement systems for condensation remains a challenge due to the complexity of the involved flow dynamics and the multitude of interacting variables[29]. These include jet spacing, nozzle to surface distance, and crossflow management, all of which can significantly influence the overall condensation efficiency[28], [32]. This literature review aims to analyze these factors, focusing on how

geometrical configurations, flow interactions, and surface properties affect the performance of small jet impingement condensers.

This review is structured to first explore the fundamental physics governing jet impingement and its relevance to condensation. Following this, the discussion will dig into key geometrical variables that influence condensation efficiency, including jet to plate distance (H/D), nozzle to nozzle spacing (S/D), jet patterns and other parameters as the area of the target plate and the temperatures. Finally, the flow characteristics of jet impingement and bubble columns will be reviewed, providing a comprehensive view of both theoretical and applied aspects of this technology.

2.3. Physics of Jet Impingement Condensation

Jet impingement is characterized by the presence of distinct flow regions free jet, stagnation, and wall jet each of which plays a crucial role in determining the overall heat and mass transfer rates[33]. In the context of condensation, these regions influence droplet formation, growth, and removal, making them key to optimizing condensation rates[34].

In the free jet region, the flow exits the nozzle with a stable velocity profile that remains largely intact until it encounters the surrounding environment, where shear layers begin to form. These shear layers develop into vortices that drive turbulence, enhancing the mixing and energy transfer processes[35]. For small-scale condensers, maintaining the integrity of the jet in this region is critical for ensuring that the subsequent impingement is effective at promoting condensation[36].

The stagnation region, where the jet hits the surface directly, has the highest local heat transfer rates due to the focused turbulent energy [31]. In a condensation scenario, this zone

is where vapor rapidly cools and condenses into droplets. The combination of high velocity and low boundary layer thickness near the stagnation point significantly accelerates the condensation process[37].

As the flow spreads radially along the surface, it enters the wall jet region, where the velocity decreases and the flow transitions from axial to radial. In condensation systems, this region is particularly important because it determines how effectively condensed droplets are transported away from the surface. If the wall jet region is well-optimized, it can lead to secondary peaks in heat transfer due to recirculation and vortex structures, which can help remove condensate and prevent flooding of the surface[38].

In multiple jet configurations, additional factors such as jet interference and crossflow interactions become critical considerations[39]. Weigand and Spring conducted a comprehensive review of such systems, focusing particularly on the effects of crossflow in confined jet arrays. Their study employed a combination of experimental methods and analytical correlations to evaluate the influence of nozzle to nozzle spacing (S/D), jet to plate distance (H/D), and crossflow on heat transfer, examining both inline and staggered jet arrangements across a broad range of Reynolds numbers. The parameters explored included S/D ratios from 0.5 to 40, H/D ratios from 0.5 to 40, and Reynolds numbers varying from 1,000 to 100,000, under varying degrees of crossflow conditions. The results indicated that closely packed jet arrays are particularly susceptible to significant crossflow effects, leading to diminished heat transfer in downstream jets. Optimal performance was achieved with moderate S/D ratios and small H/D values, minimizing crossflow disturbances. Based on these findings, the authors recommended avoiding high jet densities and suggested configurations that strike a balance between crossflow impacts and overall

system efficiency[40]. Moreover, an experimental analysis conducted by Hrycak to investigate the behavior of multiple Circular and slot jets directed onto both flat and curved surfaces, emphasizing how different geometrical configurations influence heat transfer and fluid dynamics. By varying jet to plate distances (H/D : 1 to 45) and incorporating crossflow conditions, the study revealed that crossflow has a lesser effect on slot jets than on circular jets, which experience significant performance degradation. Additionally, Hrycak highlighted the detrimental effects of spent air, particularly in closely spaced jet configurations, underscoring the importance of managing crossflow and optimizing jet spacing to enhance overall system performance [41].

Given the significance and widespread applications of jet impingement, numerous reviews and important articles have been published over the past years, as summarized in Table 1. The identified geometric variables in a multiple jet impingement system are essential in determining jet dynamics, which directly influence efficiency of heat transfer on the target plate. These factors will be addressed in detail in the next section.

Table 1 Review of single and multiple jet impingement systems

Reference and Year	Methodology	Jet Impingement Type	Parameter Analyzed and Parameter Values	Major Finding
[42] Year 1971	Comprehensive review of experimental and theoretical research	Single and Multiple Jets	Parameters Analyzed Jet Exit Velocity, Jet to Plate Distance (H/D), Reynolds Number, Turbulence Levels and Crossflow Effects	Elevated jet velocities improve heat transfer efficiency; optimal H/D ratios range between 1 and 10. Turbulence and crossflow are critical factors in performance.

Table 1. continued

			Parameter Values Pressure distribution; $450 \leq Re \leq 22,000$; $2.5 \leq U' \leq 18 \%$; Circular and slot jets; $1 \leq H/D \leq 45$	
[43] Year: 1995	Comprehensive review compiling experimental and analytical studies on single-phase liquid jet impingement	Single-Phase Liquid Jet	Parameter Analyze Flow Configuration (Submerged & Free-surface), Nozzle to Plate Distance and Jet Inclination Parameter Values Axisymmetric and planar jets; $0 \leq H/D \leq 25$; $40^\circ \leq \theta \leq 90^\circ$; Surface: roughness, curvature, motion	Submerged jets enhance turbulence and heat transfer, while free-surface jets are more influenced by gravity. Optimizing jet impingement improves cooling in high-heat flux systems. Adding a dispersed second phase in gas jets enhances heat transfer, with surface curvature impacting film thickness.
[44] Year: 2004	Review of experimental and simulation studies related to air jet impingement	Single and Multiple Jets	Parameters Analyzed Heat Transfer Coefficient, Reynolds Number, Nozzle to Plate Distance (H/D Ratio) and Turbulence Levels Parameter Values Reynolds Number: Up to 46,300; H/D Ratio: 6 to 8	Air impingement significantly reduces processing times; circular jets excel in peak heat transfer, while slot jets ensure better uniformity. Crossflow and nozzle spacing affect overall performance.
[45] Year: 2020	Review of experimental studies on air and liquid jets	Single and Multiple Inclined Jets	Parameters Analyzed Inclination Angle, Reynolds	lower inclination angles ($<30^\circ$) lead to a significant drop in heat transfer

Table 1. continued

	impinging on flat surfaces		Number and Jet to Plate Spacing Parameter Values Reynolds Number: 1,500 to 52,000; Inclination Angle: 15° to 90° and Jet to Plate Spacing: 2 to 12; Liquid and air jets	efficiency due to the disengagement of fluid from the surface. Additionally, the maximum heat transfer point shifts towards the uphill side as the jet angle decreases, with liquid jets exhibiting a more significant displacement compared to air jets.
[29] Year: 2023	Comprehensive review of experimental and numerical studies	Multiple Air Jets	Parameters Analyzed Reynolds Number, Jet to Plate Spacing(H/D), Inclination Angle, Spacing (S/D), Crossflow Effects Parameter Values Reynolds Number: 1,000 to 100,000 H/D Ratio: 0.5 to 40 S/D Ratio: 2 to 10	The review highlights that crossflow effects degrade downstream heat transfer, and staggered jet configurations can enhance overall performance by increasing turbulence. Optimal heat transfer is achieved with moderate H/D ratios (2 to 3) and S/D ratios (5 to 6).

2.4. Influence of Geometrical Variables and other parameters on condensation efficiency

The efficiency of small jet impingement condensers is significantly affected by the precise selection and regulation of geometric variables, including the jet to plate distance

(H/D), nozzle to nozzle spacing (S/D), and jet configurations (inline versus staggered)[28], [32]. These elements are critical in sustaining high condensation rates by influencing the uniformity of surface coverage and controlling flow interactions. Furthermore, additional key geometric factors, such as the geometry and motion of the target surface, jet angle and temperature, as well as Mach number, and the area of the target surface, are also considered[40].

2.4.1. Jet to Plate Spacing (H/D):

The distance between the nozzle and the target surface plays a fundamental role in determining the effectiveness of condensation [46], [47]. Maintaining an optimal H/D ratio ensures that the jet retains sufficient momentum to drive effective condensation while minimizing the adverse effects of crossflow [48]. Studies indicate that an H/D ratio of 2 offers the best performance across various applications, balancing jet momentum with vortex formation near the impingement point. Choo and Kim's study examined how jet-to-target spacing affects heat transfer performance in a system with multiple rows of impinging air jets. Using experimental methods and infrared thermography, they analyzed various configurations, focusing on jet to target spacing (H/D), nozzle to nozzle spacing, and jet array arrangements (inline and staggered), with Reynolds numbers between 5,000 and 25,000. The study found that smaller jet-to-target spacings ($H/D = 2$ to 4) improved heat transfer due to increased turbulence and stronger impingement. However, larger spacings led to reduced performance, especially with high crossflow interference. The research identified an optimal jet-to-target spacing where heat transfer is maximized while minimizing negative crossflow effects [39]. Another an experimental study by Weihong Li et al. conducted to investigate the heat transfer behavior of an inline impinging jet array,

focusing on low jet-to-target distances (H/D) between 0.75 and 3. By means of a transient liquid crystal method, the study evaluated local and average heat transfer coefficients across a range of Reynolds numbers starting from 5,000 to 25,000, while varying jet to target distances, jet spacing, and jet angle angles (0° to 40°). The findings indicated that lower H/D ratios improved heat transfer rates due to increased turbulence and closer jet proximity to the target surface, while performance decreased as H/D increased. The study found that the optimal heat transfer efficiency occurred at a jet angle of 20° , after which performance declined due to increased impingement distance [48]. However, research by [49] demonstrates that for $0 \leq H/D \leq 1.2$, the heat transfer is less effective compared to $H/D = 2$, suggesting that there is an ideal jet-to-target distance where heat transfer efficiency is maximized.

The studies showed that heat transfer on an impinging surface is optimized when $H/D = 2$. However, when this ratio is significantly low ($0 < H/D < 1.2$), the flow becomes dominated by crossflow, leading to a decrease in heat transfer efficiency. A summary of various studies examining the jet to plate distance is presented in Table 2.

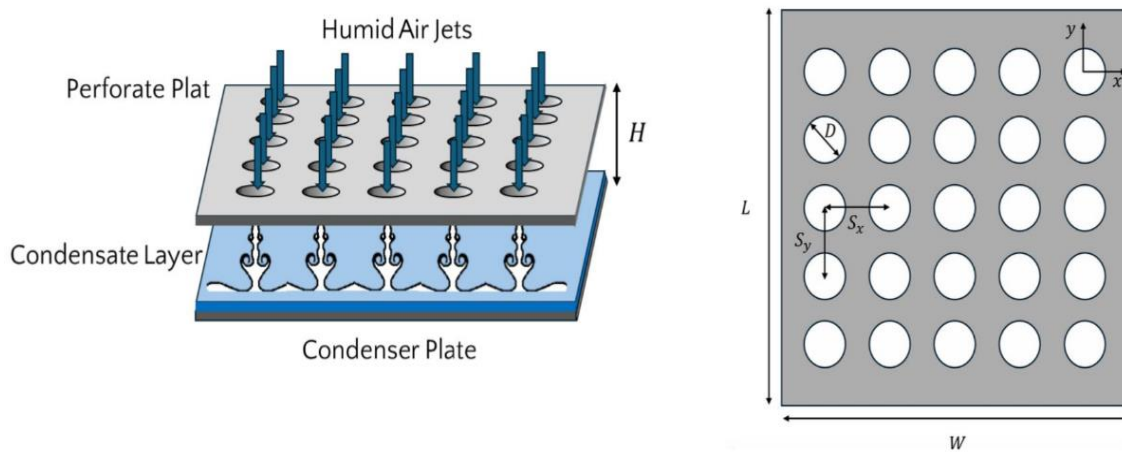


Figure 4 Jet impingement geometric parameters.

Table 2 Review of nozzle to plate spacing

Reference Year	Methodology	Jet Impingement Type	Parameter Analyzed and Parameter Values	Major Finding
[50] Year: 2008	Experimental study focused on hole spacing effects in jet array impingement using various Reynolds and Mach numbers	Array of circular impinging jets	Parameter Analyzed Jet to Plate Spacing (H/D), Jet to Jet Spacing (S/D), Reynolds Number and Mach Number Parameter Values H/D Ratio: 3; Jet Spacing: 8D and 12D and Reynolds Number: 8,200 to 30,500	The study found that an H/D ratio of 3 provided the best balance, while lower ratios increased localized heat transfer but also crossflow, reducing efficiency. Larger ratios led to less effective heat transfer.
[51] Year: 2010	Analysis of flow characteristics and heat transfer by experimentation and numerical methods	Circular Jet Array	Parameter Analyzed Jet to Plate Distance (H/D), Reynolds Number and Crossflow Patterns Parameter Values H/D Ratios: 3, 4, 5 Reynolds Number: 35,000	The study found that for configurations with significant crossflow, smaller H/D ratios (H/D = 3) provided optimal heat transfer efficiency, particularly in the initial jet rows. Increasing the H/D ratio reduced heat transfer efficiency due to weaker impingement and increased flow divergence.
[52] Year: 2012	Experimental study utilizing the Taguchi method for design optimization	Co-axial and Circular Jets	Parameter Analyzed Jet to Plate Distance (H/D), Reynolds Number, Surface	The study found that the jet to plate distance (H/D) significantly affects heat transfer. The

Table 2. continued

			Roughness and Jet Type Parameter Values H/D Ratios: 2, 6, 8, 10; Reynolds Numbers: 20,000, 40,000	Nusselt number decreases as the H/D ratio increases, with the optimal H/D found to be 2 for maximum heat transfer
[53] Year: 2014	Experimental and numerical investigation using elliptic and rectangular jets. Correlations for average Nusselt values were obtained by statistical regression analysis.	Multiple jets (Elliptic and Rectangular)	Parameter Analyzed Jet to Plate Distance (H/D), Reynolds Number and Aspect Ratio Parameter Values H/D = 2 and 10, Re = 2000 to 10,000	The study discovered that the highest average Nusselt number occurred at H/D = 2 and Re = 10,000 for elliptic jets, with jet geometry and aspect ratio significantly affecting heat transfer rates, particularly at low H/D values.
[54] Year: 2015	Experimental study using stereoscopic PIV and thermosensitive liquid crystal (TLC) techniques	Square array of multiple circular jets	Parameter Analyzed Jet to Plate Spacing (H/D), Jet to Jet Spacing (S/D), Flow Configuration and Nusselt Number Parameter Values H/D Ratio: 2 to 6; S/D Ratio: 4 to 6	Smaller H/D ratios enhance vortex formation and heat transfer, while jet interactions and crossflow strongly impact overall heat transfer

2.4.2. Nozzle to nozzle Spacing (S/D):

The spacing between jets is crucial for achieving uniform coverage and preventing negative interactions. Because of the intense interactions between the wall jets, this geometrical parameter impacts the upstream Interaction among the jets before they impinge upon the target surface and extend along the wall. The jet array can be configured in two

ways: irregular, where the spanwise (S_y) and streamwise (S_x) distances differ, and regular, where $S_x = S_y = S$ [50], [55]. When S/D is set between 2 and 3, the system achieves optimal performance by striking a balance between minimizing upstream jet interference and maximizing surface coverage [29]. Vadiraj Katti and S.V. Prabhu conducted an experimental study to investigate the influence of spanwise nozzle to nozzle spacing (S/D) on the distribution of local heat transfer in in-line arrays of circular jets. The research varied the spanwise spacing between 2D, 4D, and 6D while maintaining a constant streamwise spacing of 5D and explored different jet to plate distances (H/D). They found that smaller S/D ratios (2D) resulted in strong jet interactions, leading to higher heat transfer near the first rows but decreased efficiency further downstream due to crossflow. In contrast, larger S/D ratios (6D) provided more uniform heat transfer but with lower peak performance. The optimal configuration was identified at $S/D = 4D$, which balanced heat transfer uniformity and overall efficiency [56]. While Badra et al. employed numerical simulations to assess the transient heat transfer resulting from circular jet arrays directed onto a moving plate. They focused on the impact of nozzle spacing (S/D) on heat transfer rates. The findings showed that when S/D is small (e.g., $S/D = 2$), significant interactions between adjacent jets occur, leading to additional peaks in the Nusselt number due to flow interactions. As S/D increases (e.g., $S/D \geq 10$), the jets operate more independently, resembling single jet behavior with more uniform heat transfer distribution. The study identified that an optimal nozzle spacing of $S/D = 5$ balances minimizing jet interactions while maintaining effective heat transfer performance. The studies show that optimal heat transfer occurs at a nozzle to nozzle spacing (S/D) between 2 and 3, where turbulence and surface coverage are balanced. Smaller S/D values intensify turbulence but can cause excessive crossflow,

weakening jet strength and reducing heat transfer. Larger S/D values lead to reduced jet interactions and less effective surface coverage, resulting in decreased heat transfer efficiency [29]. A summary of various studies examining the nozzle to nozzle (S/D) distance is presented in Table 3.

Table 3 Review of nozzle to nozzle spacing

Reference And Year	Methodology	Jet Impingement Type	Parameter Analyzed and Parameter Values	Major Finding
[57] Year: 1994	Experimental	Confined, isothermal axisymmetric air jets	Parameter Analyzed Jet spacing (S/D), H/D, Reynolds Number Parameter Values S/D: 4, 6, 8; H/D: 0.25, 1.0, 6.0; Re: 3500-20,400	Smaller S/D values, like 4, provided better heat transfer and more uniform coverage. As S/D increased to 8, jet interactions weakened, reducing heat transfer. The best performance was at S/D = 4, especially with low H/D values, due to stronger jet interactions and vortex formation.
[58] Year: 2011	Experimental and numerical	Multiple jet systems (Array of slot nozzles)	Parameter Analyzed Nozzle spacing (S/D), H/D, Reynolds Number Parameter Values S/D: 6-18, H/D: 3-10; Re: up to 100,000	At low S/D values, strong jet interactions cause instability and uneven heat transfer. As S/D increases, heat transfer stabilizes, with the best results at S/D = 10, where jet interactions are balanced.

Table 3. continued

[39] Year: 2015	Experimental	Square array of circular jets	Parameter Analyzed Jet spacing (S/D), H/D, Reynolds Number Parameter Values S/D: 4-6; H/D: 2-6; Re: 3000	Smaller S/D values, like 4, result in stronger jet interactions and more vortex activity, boosting heat transfer. As S/D increases to 6, these interactions weaken, reducing heat transfer. The study found S/D = 4 to be the best for maximizing heat transfer.
[59] Year: 2017	Numerical	Hexagonal array of impinging jets	Parameter Analyzed Nozzle spacing (S/D), H/D, Reynolds Number Parameter Values S/D: 2; H/D: 4; Re: 20,000	They found that at S/D = 2, strong jet interactions create flow irregularities and vortices, which improve heat transfer. The frequent jet interactions lead to uneven but stable cooling, making it effective for high-performance cooling applications.

2.4.3. Jet Pattern (Inline vs. Staggered):

The arrangement of jets, whether inline or staggered, significantly impacts the uniformity of condensation. In a staggered arrangement, jet interactions tend to be more complex, often leading to more uniform coverage and enhanced mixing. In contrast, inline configurations may result in less uniformity due to more predictable and aligned flow patterns. Therefore, the choice of jet arrangement plays a vital role in concluding the

consistency and effectiveness of condensation in applications such as cooling and heat transfer systems[40]. These differences become notable when the nozzle to nozzle spacing is set to $S_x/D = 5$ and $S_y/D = 4$, with separation distances (H/D) of 2 and 3, and elevated crossflow. Under this structure, the inline array shows superior performance with respect to average heat transfer rates [42]. Xing et al. carried out experimental and numerical analysis comparing the heat transfer performance of inline and staggered jet patterns in impinging jet arrays. They found that the inline design consistently achieved higher heat transfer coefficients compared to the staggered pattern. This was because the inline arrangement better protected downstream jets from crossflow, leading to more uniform and efficient cooling. In contrast, the staggered pattern allowed greater crossflow interference, resulting in less consistent heat transfer. The optimal performance was observed with the inline pattern at a jet to plate spacing (H/D) of 3[43]. However, an experimental study by Barbosa et al. investigated the effects of jet patterns (inline vs. staggered) in a multiple air jet impingement system using the Taguchi method and ANOVA. The study found that the staggered jet pattern slightly enhanced heat transfers due to stronger crossflow interactions between upstream and downstream jets. However, the impact of jet pattern was less significant compared to other parameters like jet spacing (S/D) and jet to plate distance (H/D). The optimal configuration for maximizing heat transfer was identified as $S/D = 3D$ and $H/D = 2D$, regardless of the jet pattern used[44]. The results in table 4 show that for $S/D \geq 3$ and $H/D = 2$, the staggered pattern boosts heat transfer due to better mixing from jet induced crossflow. However, for $S/D < 3$ and $H/D \geq 3$, weaker jet momentum and increased crossflow reduce heat transfer, making the inline configuration more effective

by protecting downstream jets. Balancing these variables is key to maintaining optimal turbulence for heat transfer.

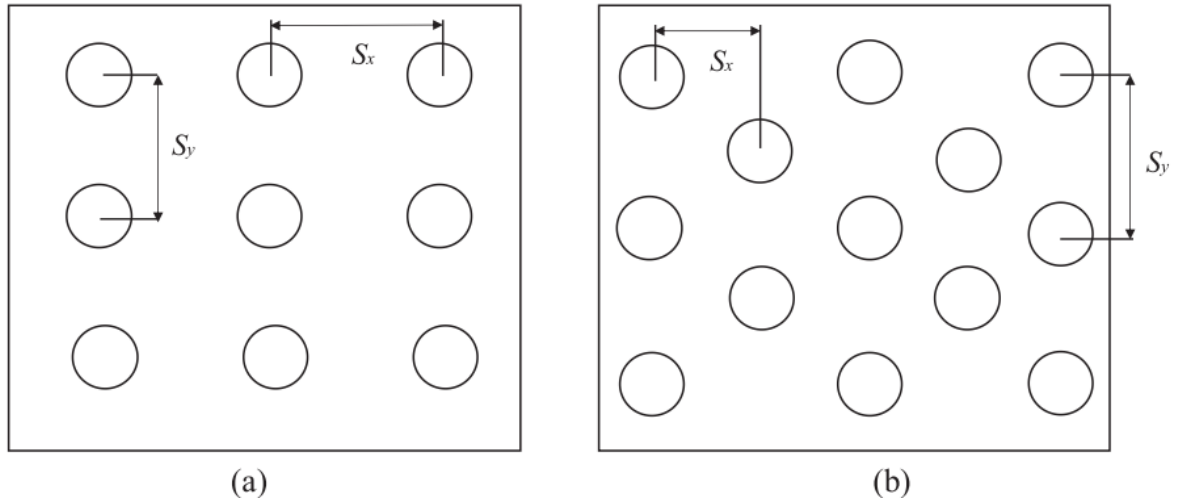


Figure 5 Jets Pattern (a) inline, (b) staggered [29].

Table 4 Review of Jet Pattern

Reference And Year	Methodology	Jet Impingement Type	Parameter Analyzed and Parameter Values	Major Finding
[49] Year: 1981	Experimental and Theoretical	Inline and Staggered Arrays of Circular Jets	Parameter Analyzed Jet Pattern (Inline vs. Staggered), Jet Spacing (S/D), Crossflow effects Parameter Values S/D: 5-15; H/D: 1-3; Re: 2,500-70,000	The researchers found that the inline pattern achieved higher Nusselt numbers, especially under low crossflow, while the staggered pattern suffered reduced heat transfer due to crossflow. The inline configuration handled crossflow better, resulting in more consistent heat transfer.

Table 4. continued

[61] Year: 1996	Experimental	Inline and Staggered Arrays of Circular Jets	Parameter Analyzed Jet Pattern (Inline vs. Staggered), Nozzle Spacing (S/D), Crossflow Effects Parameter Values S/D: 8; H/D: 1, 2, 4; Re: 10,000 - 40,000	They found that the inline jet pattern provided more consistent and higher heat transfer, while the staggered pattern was more affected by crossflow. The inline configuration was more effective overall, especially at lower crossflow conditions.
[39] Year: 2015	Experimental	Inline and Staggered Arrays of Multiple Air Jets	Parameter Analyzed Jet Pattern (Inline vs. Staggered), Nozzle Spacing (S/D), Jet to target Spacing (H/D) Parameter Values S/D: 2-5; H/D: 2-4; Re: 5000-25000	The staggered pattern had better heat transfer in the front rows but was impacted by crossflow in the rear rows, while the inline pattern performed better overall, especially at S/D = 3 and H/D = 2.
[62] Year: 2020	Experimental and Numerical (CFD)	Inline and Staggered Circular Jets	Parameter Analyzed Jet Pattern (Inline vs. Staggered), Nozzle Spacing (S/D), Nozzle to target Spacing Parameter Values (H/d) S/D: 1.4-2.9; H/d: 2-12; Re: 5000-20000	The inline pattern provided more consistent and uniform heat transfer, while the staggered pattern caused more turbulence and non-uniform cooling, especially near the cylinder edges. The inline configuration was more effective for uniform cooling, particularly at lower H/D values and smaller S/D spacing.

2.5. Effect of Area of the Target Surface:

In multiple jet impingement, the characteristics of the target surface play a critical role in determining heat transfer efficiency. The surface area can be modified by introducing geometric features such as roughness, dimples, or varying curvatures, which directly affect local heat transfer by altering turbulence, flow separation, and the boundary layer thickness [40]. Various studies have explored these effects under different conditions, providing insights into the optimal configurations for enhanced cooling or heating performance. Caliskan and Baskaya conducted an experimental study to analyze the impact of ribbed surfaces on heat transfer in jet impingement setups. They compared smooth surfaces with V-shaped and convergent-divergent (CD) ribs, focusing on how increasing the target surface area through rib configurations influences heat transfer. The findings indicated that the V-shaped ribs provided the best enhancement, with average Nusselt numbers increasing by 4% to 26.6% compared to smooth surfaces. This improvement was as a result of the increased surface area and turbulence generated by the rib geometry. The enhancement was most significant at smaller jet to plate spacings ($H/D = 2$) and higher Reynolds numbers ($Re = 10,000$) [63]. Additionally, experimental and numerical analysis conducted by Brakmann et al. to examine how heat transmission in an impinging jet array is affected by cubic micropin fins. The study showed that adding these micro pin fins increased the target surface area by 150%, leading to a significant improvement in overall heat transfer, with an increase of around 134% to 142%. Although the Nusselt number slightly decreased due to the additional surface roughness, the larger area compensated for this reduction. The micro pin fins were particularly effective in crossflow conditions, where they disrupted

flow paths and enhanced cooling performance. Table 5 provides a summary of several studies investigating the impact of the area of the target surface.

Table 5 Reviews of the impact of the area of the target surface

Reference And Year	Methodology	Jet Impingement Type	Parameter Analyzed and Parameter Values	Major Finding
[64] Year: 2003	Experimental investigation using impingement flat wall cooling with square hole arrays and rib turbulators	Co-flow and Cross-flow	Parameter Analyzed Heat transfer coefficient, pressure loss Parameter Values Impingement gap (H/D): 2.75, Hole diameter: 4.66	They found that slotted ribs increased heat transfer by 20% in the downstream region, while smooth ribs in co-flow reduced pressure loss by 25%. However, none of the rib configurations improved overall surface heat transfer compared to a smooth wall.
[65] Year: 2010	Experimental	Multi-jet array impinging on flat and dimpled plates	Parameter Analyzed Surface type (Flat vs. Dimpled), Jet to plate spacing (H/D), Crossflow scheme Parameter Values Flat and dimpled plates, H/D = 3, 4, 5; Re = 15,000-35,000	Resulting that the dimpled surface improved the effective target area by 26.4%, enhancing heat transfer compared to flat surfaces. The greatest improvement, with a 12.3% increase in the Nusselt number, occurred at narrow jet to plate spacing (H/D = 3) and low crossflow conditions.

Table 5. continued

66 2017	Experimental & Numerical	Multi-jet array impinging on a small-scale cylinder roughness surface	Parameter Analyzed Target surface roughness, Cylinder height (H/D), Reynolds number Parameter Values H/D: 0.125, 0.188, 0.250; Re: 900-11,000	Finding that increasing surface area with cylinder roughness arrays (up to 94% more) improved heat transfer, especially at higher Reynolds numbers. However, too much roughness reduced performance due to added thermal resistance. The best heat transfer occurred at a moderate roughness height of H/D = 0.188
[67] Year: 2021	Numerical Simulation	Multiple jets impinging on a moving curved surface	Parameter Analyzed Target surface area, Nozzle spacing (S/D), Jet inlet velocity, Jet angle Parameter Values S/D: 2-10; H/D: 2-10; Re: 4,337-21,685	The study found that increasing the target surface area by adjusting the surface curvature and jet spacing reduced drying energy consumption. The optimized design showed that larger nozzle to nozzle spacing (S/D = 10) and specific surface curvature improved overall energy efficiency, particularly for drying applications, by enhancing heat and mass transfer.

2.6. Effect of Temperatures

Temperature is a key factor affecting the condensation process in jet impingement systems[45]. The variation between the vapor and surface temperatures controls the condensation rate, with larger temperature gradients resulting in higher condensation efficiency[46]. However, maintaining an optimal temperature is essential, as excessively low target surface temperatures can lead to excessive droplet formation, which can accumulate and hinder heat transfer. Conversely, insufficient cooling can reduce the overall condensation rate by limiting the rate at which vapor condenses[47]. Experimental findings have indicated that controlling the temperature gradient is particularly important in small-scale systems where rapid cooling is necessary to maximize heat transfer. Furthermore, the effectiveness of the condensation process is closely tied to the uniformity of temperature across the impingement surface, making temperature control a key aspect of optimizing jet impingement condensers[48].

In addition to the overall temperature gradient, the temperature of the target surface plays a significant role in determining condensation efficiency. Studies have shown that a cooler target surface facilitates faster phase change from vapor to liquid, leading to enhanced droplet formation and improved condensation rates. However, maintaining the surface temperature within an optimal range is crucial; if the surface is too cold, droplet coalescence and flooding may occur, reducing the heat transfer efficiency. This balance is particularly important in systems where the target surface temperature must be controlled to avoid the formation of excessive frost or liquid buildup, which could impede the overall system performance[49].

The temperature of the evaporated vapor from the evaporation system is another key factor that influences condensation. When vapor is generated at higher temperatures, the driving force for condensation increases caused by the significant temperature difference between the vapor and the cooled target surface [49]. However, excessively high vapor temperatures may introduce challenges in maintaining a stable condensation process, as the heat load on the impingement surface increases[50]. In evaporation and condensation systems, precise control over both the vapor temperature and the target surface temperature is essential to achieve consistent condensation rates[49]. Table 6 summarizes several studies investigating the effect of temperature.

Table 6 Reviews of the impact of the temperature

Reference And Year	Methodology	Jet Impingement Type	Parameter Analyzed and Parameter Values	Major Finding
[73] Year: 1995	Experimental study using infrared thermography to measure adiabatic wall temperature	Single circular jet	Parameter Analyzed Mach number, jet to plate distance, nozzle geometry, recovery factor Parameter Values Mach number: 0.4–0.95; H/D: 2–20; Jet Temperature: 10°C above ambient in some tests	At Mach numbers above 0.7, temperature instability occurs with azimuthal fluctuations and lobed structures. Two temperature drops are seen at $H/D < 6$ due to vortex ring breakup.
[74] Year: 2013	Computational Fluid Dynamics (CFD) simulation using FLUENT, validated against	Single and multiple circular jets	Parameter Analyzed Reynolds number, jet to plate distance, jet spacing, temperature	Lower H/D (2) shows concentrated heat transfer with localized temperature peaks. As H/D

Table 6. continued

	experimental data		Parameter Values Reynolds number: 5,000–66,000; Jet to Plate Distance (H/D): 2–8; Jet Spacing (S/D): 2–10; Jet Temperature: 400°C	increases, temperature distribution becomes more uniform with fewer fluctuations.
[75] Year: 2016	Experimental study using infrared thermography and thin metal foil technique	Single circular, square, and triangular jets	Parameter Analyzed Reynolds number, jet to plate distance, jet temperature, nozzle shape Parameter Values Reynolds number: 5,000–23,000; Jet Temperature: 70°C–175°C; H/D: 1–10	Jet temperature had minimal impact on heat transfer. Heated jets produced slightly lower Nusselt numbers than ambient jets, especially in the stagnation region ($r/de < 1$).
[76] Year: 2016	Experimental study using stereoscopic PIV, Laser Doppler Velocimetry (LDV), Cold Wire Thermometry (CWT)	Single circular heated jet	Parameter Analyzed Reynolds number, jet exit temperature, jet to plate distance, Nusselt number Parameter Values Reynolds number: 60,000; Jet exit temperature: 130°C; Jet to plate distance (H/D): 3	Higher jet exit temperatures (130°C) create strong thermal gradients near the stagnation point. Significant temperature fluctuations occur near the wall, reaching up to 113°C, causing localized cooling effects.
[77] Year: 2019	Experimental study using infrared thermography and particle image velocimetry (PIV)	Single compressible jet	Parameter Analyzed Mach number, injection diameter, injection-to-plate distance, temperature	At higher Mach numbers (0.8), temperature becomes unstable, causing significant variation across the plate. Lower

Table 6. continued

			Parameter Values Mach number: 0.3–0.8; Injection diameter: 6–15 mm; Injection-to-plate distance (H/D): 1–8; Jet Temperature: Ambient (20°C)	H/D values result in higher temperature gradients and localized cooling. At higher H/D, ambient air raises static temperature, making wall temperatures exceed the jet temperature.
--	--	--	---	---

2.7. Flow Dynamics

The flow dynamics of jet impingement are crucial in determining heat transfer performance, particularly influencing condensation rates in small jet impingement condensers. Key factors such as crossflow management, and the generation of secondary vortices must be carefully controlled to enhance heat transfer and optimize overall condensation efficiency[29].

Crossflow, resulting from the accumulation of spent air from upstream jets, can either enhance or diminish heat transfer performance, thereby affecting condensation efficiency. In scenarios with significant crossflow, the incoming jets may lose momentum before reaching the target surface, leading to reduced condensation efficiency. Conversely, when crossflow is controlled, it can enhance heat transfer by promoting a more uniform distribution of vapor across the surface, which supports improved droplet formation and overall condensation effectiveness[34].

Secondary vortices, generated through the interaction between wall jets and surrounding flow, play a significant role in enhancing local turbulence. These vortices help in removing condensed droplets, thereby mitigating surface flooding and contributing to better heat

transfer performance and more efficient condensation[51]. Experimental data indicates that optimizing the jet to surface distance (H/D) and the spacing between jets (S/D) significantly influences the formation and distribution of vortices. These optimizations enhance heat transfer performance by promoting more effective fluid mixing and improved impingement dynamics, ultimately leading to better condensation outcomes.[52].

Numerical studies using turbulence models such as RANS and LES validate that secondary vortices significantly enhance the overall heat and mass transfer rates. Simulations indicate that at an H/D ratio of 2, well-developed vortices near the surface promote higher local heat transfer rates, which could be directly correlated to improved condensation performance in relevant applications[53]. These findings underscore the critical importance of accounting for flow dynamics, including jet spacing and interactions, when designing jet impingement systems aimed at optimizing condensation performance[54].

The intricate flow dynamics of jet impingement, including the management of crossflow and the generation of secondary vortices, play a pivotal role in optimizing heat and mass transfer, directly impacting condensation performance. These insights emphasize the importance of controlling fluid interactions and turbulence to achieve efficient heat exchange. Integrating jet impingement condensers (JIC) with two-phase bubble column systems can further enhance overall system efficiency by leveraging the bubble column's ability to maximize vapor generation and improve humidification. This combination allows both technologies to work together effectively, resulting in improved heat and mass transfer performance.

2.8. Two-Phase Bubble Columns

Two-phase bubble columns are commonly used in industries because they offer advantages in both operation and system design[55]. An experimental analysis was performed to investigate the process of humidification by air passing through saltwater in a bubble column setup. The results demonstrated that increasing the temperature of water and controlling the velocity of air significantly impacts the vapor content difference and humidification efficiency. For example, at a water temperature of 75°C and an air velocity of 7.5 m/s, the maximum vapor content difference achieved was approximately 222 g/kg of dry air. The study highlights that higher water temperatures lead to more effective vapor production, and optimizing the headwater difference in the bubble column further enhances the humidification process[56].

An experimental study carried out by S.A. El-Agouz to evaluate the impact of operating conditions on the vapor content in a bubble column humidifier used in a desalination process. He explored variables such as temperature of water, seawater level, and airflow rate. The findings demonstrated that raising the water's temperature while optimizing the airflow rate significantly enhances vapor content in the system. For example, at a water temperature of 86°C and an airflow rate of 14 kg/h, the system reached a maximum productivity of 8.22 kg/h. Additionally, the research indicated that while water temperature and airflow rate had a substantial influence on vapor content, the seawater level had a relatively minor effect.

Additionally, experimental research was carried out by Ghazal et al. to evaluate the performance of a solar humidifier integrated into a humidification-dehumidification (HDH) system. The study focused on the humidification process where air is introduced as

bubbles into a water-filled solar collector. The results indicated that as air bubbles rise through the water, the temperature of the air approaches the water temperature, significantly increasing its vapor-carrying capacity. The air exits the system with nearly 100% relative humidity. Introducing regenerative sieves in the bubble column reduced bubble coalescence, maintained smaller bubble sizes, and enhanced heat and mass transfer. This led to a higher vapor content in the air, with a notable 32% increase in absolute humidity under optimal conditions when using a reflective mirror setup.

CHAPTER 3

METHODOLOGY

3.1. INTRODUCTION

This study will involve both experimental and numerical investigations to analyze various parameters of a jet impingement condenser (JIC) and their effects on the condensation rate.

3.2. EXPERIMENTAL METHODOLOGY

The experimental study analyzes various parameters of a jet impingement condenser (JIC) and their effects on the condensation heat and mass transfer rates. The experimental setup illustrated schematically in Figures 6-8, includes two stages of bubble column humidifiers and the JIC assembly. Two water baths (Julabo_CP-900F) are used to heat the water in the bubble columns by circulating heated water through coils placed in each column. Each heater is assigned to one column, with independent temperature control ranging from 45°C to 80°C. Column 1 was maintained at a higher temperature than Column 2 to ensure that the relative humidity of air transported to the JIC is 100% at all times. This will be further illustrated later on in this work.

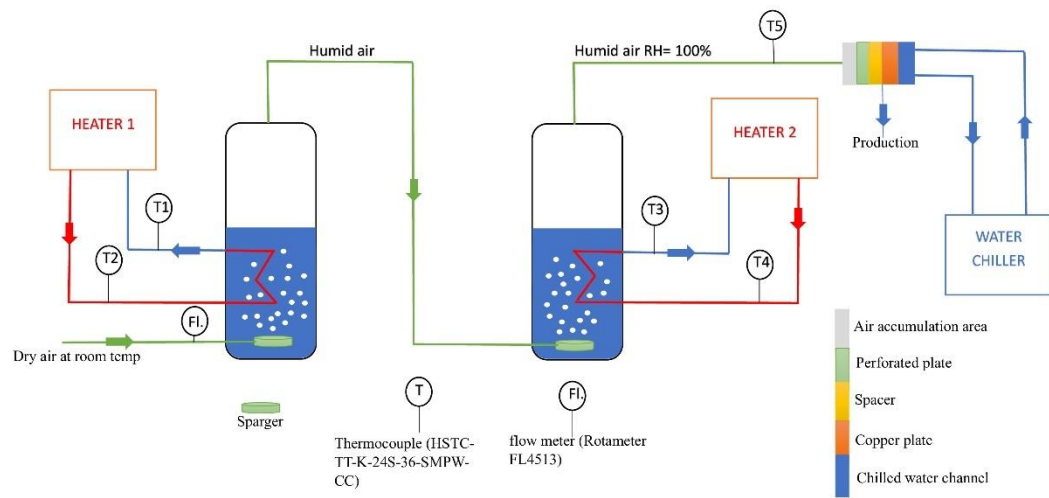


Figure 6. Two phases bubble column and JIC

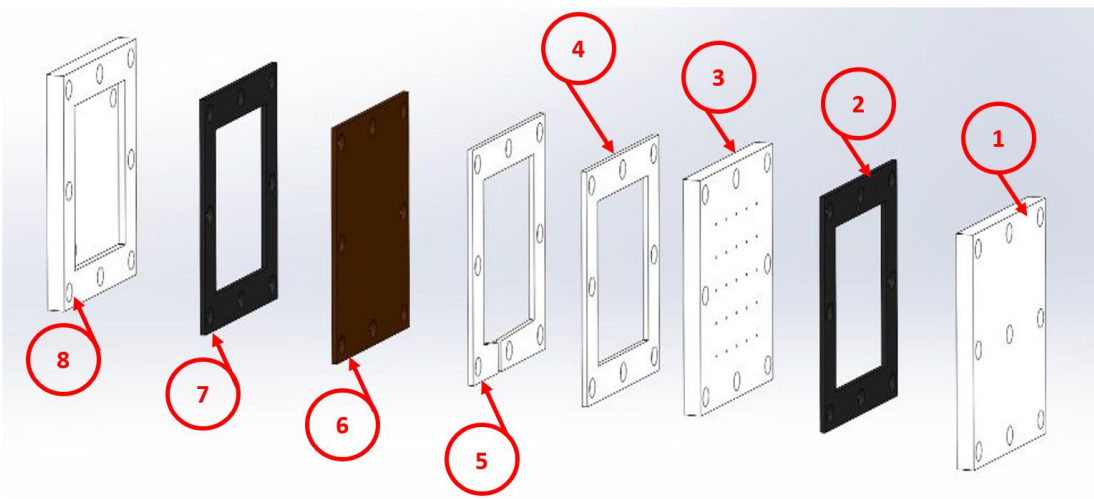


Figure 7. Jet impingement condenser model consists of 1) Air accumulation area, 2) Gasket, 3) Perforated plate, 4) Spacer, 5) Spacer 6) Copper plate, 7) Gasket, 8) Chilled water channel.

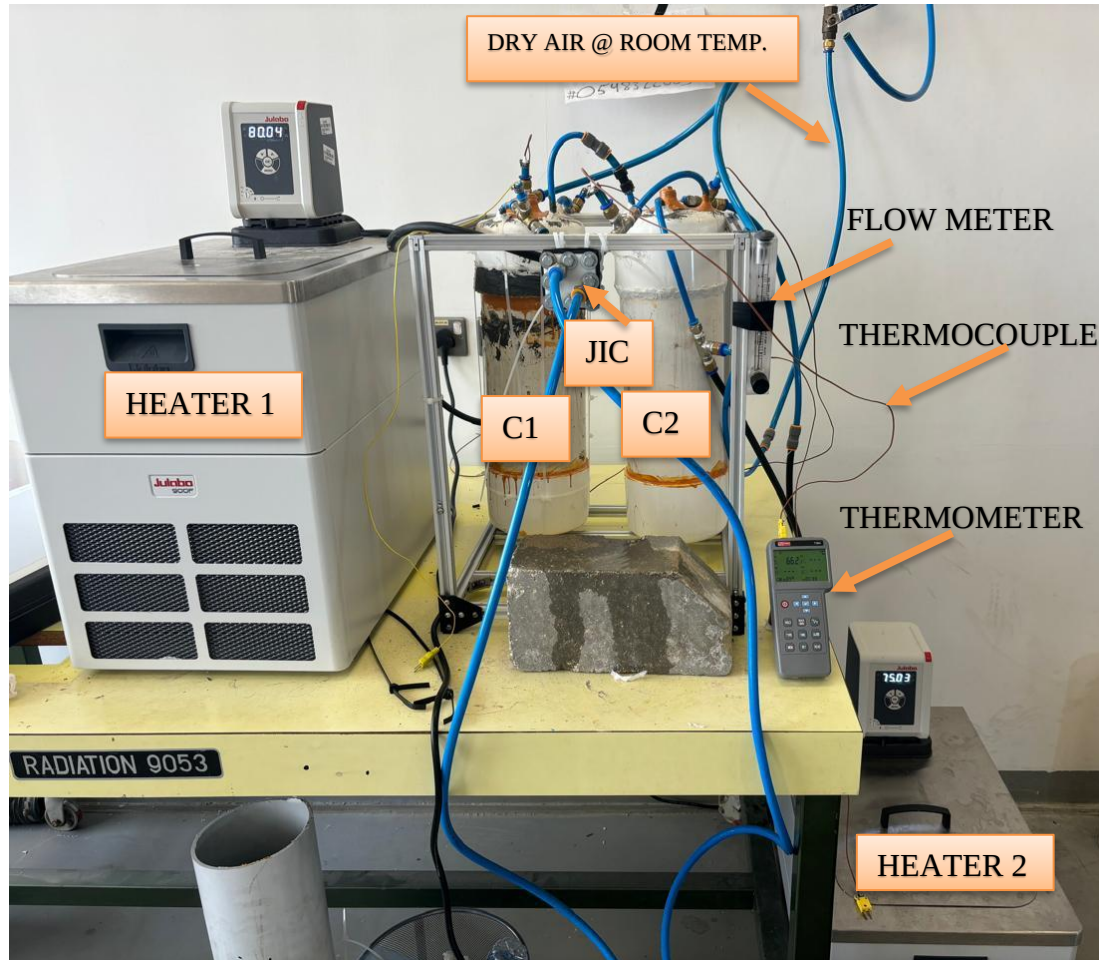


Figure 8 Experimental Setup

The water level in each column is monitored by measuring the weight of the columns. The empty columns each have a mass of 4.5 kg. When filled with water, Column 1 has a mass of 9 kg, while Column 2 has a mass of 6.5 kg. This method of monitoring the water level by tracking the mass ensures an accurate and non-intrusive way to observe the water levels during the experiment. Whenever the water level becomes low due to evaporation, it is replenished with DI water to maintain a constant level.

Dry air is supplied to Column 1 at a flow rate of 2.5 CFM, which was measured by an air flow meter (Rotameter FL4513) to an accuracy of ± 0.2 CFM. The dry air is supplied to the bottom of the bubble column through a circular sparger with a diameter of 125 mm (porous

stone bubbler). This air bubbling increases turbulence in the heated water and increases the effective surface area at which evaporation occurs, thereby enhancing evaporation rate. The humidified air produced in Column 1 is then directed to a second sparger placed at the bottom of Column 2, where it bubbles through the heated water. Because Column 2 is at lower temperature than Column 1, excess water vapor is condensed in the second column which ensure a saturated condition at the exit of the second column. The system is designed to potentially achieve up to 100% relative humidity, verified by measuring the dry and wet bulb temperatures at the outlet of Column 2 using a thermocouple (HSTC-TT-K-24S-36-SMPW-CC) and a thermometer (SDL 200). The highly humid air leaving Column 2 is directed to the JIC for condensation.

The JIC consists of several components: an air cavity, a perforated plate, a spacer, a copper plate, and a water cavity, as depicted in Figure 7. The humid air passes through a gas cavity with an internal dimension of $35 \times 35 \times 4 \text{ mm}^3$ and exits through the perforated plate. The perforated plate consists of 25 in-line drilled apertures (5×5 array), each with a diameter of 1 mm. The spacer creates a gap between the perforated plate and the target condenser surface, with a 4 mm exit at the bottom, where condensation is collected for further analysis. The entire assembly is made of polyethylene thermoplastic (HDPE) which provides inexpensive and lightweight advantages of construction. The target condenser surface is a copper plate, cooled by circulating water at 5°C through a HDPE water cavity, which also has an internal dimension of $35 \times 35 \times 4 \text{ mm}^3$. The cooling water is maintained at this temperature by a water chiller (Accel 500 LC).

Table 7 shows the different experimental configurations that was studied in this work. Each configuration is characterized by a constant jet-to-plate distance, condenser plate

area, and nozzle-to-nozzle distance. In each configuration, the temperature of the water in Column 1 is kept constant, starting from 80°C, while the temperature in Column 2 is gradually reduced from 75°C to 40°C in steps of 5°C. Then, the Column 1 temperature was kept at a lower temperature, say 75°C, and Column 2 temperature varied from 70°C to 40°C. The procedure was repeated until Column 1 temperature was 45°C and Column 2 temperature was 40°C, comprising a total of 24 experiments in each configuration.

Table 7 Summary of the experimental configurations

Configuration	Jet to Plate Distance (H/D)	Target Plate Area [mm*mm]	Water Temperature [°C]	Nozzle to Nozzle Spacing
First	2	35 × 35	45 ⁰ C to 80 ⁰ C	7
Second	2	70 × 70	45 ⁰ C to 80 ⁰ C	14
Third	14	70 × 70	45 ⁰ C to 80 ⁰ C	14

Condensation was continuously collected in a container, and production was measured at 5-minute intervals using a lab scale (B095YL6Y92). These measurements were recorded for up to 20 minutes to ensure steady production. Once a linear accumulation of condensate mass was observed with time, the condensation can be confirmed to reach a steady state condition. To calculate the condensation mass flow rate, Equation 1 was used.

$$\dot{m} = \frac{\Delta m}{\Delta t} \quad (\text{Eq.1})$$

The mass transfer flux was also calculated as the condensation mass flow rate per unit condenser area according to Equation 2.

$$\dot{m}'' = \frac{\dot{m}}{A} \quad (\text{Eq.2})$$

Where A is the condenser surface area. The total heat transfer flux was approximated as the latent heat transfer flux. This assumption is valid given the value of sensible heat transfer rate is very small compared to the latent heat transfer rate [57]. The heat transfer flux was calculated using Equation 3.

$$q'' = \dot{m}'' h_{fg} \quad (\text{Eq.3})$$

Where h_{fg} is the latent heat of vaporization. Accordingly, the Nusselt number was obtained using Equation 4.

$$Nu = \frac{q'' D}{k (T_\infty - T_s)} \quad (\text{Eq.4})$$

Where D is the aperture diameter, k is the thermal conductivity of the humid air at the film temperature. T_∞ and T_s are the temperature of the incoming humid air jet (dry temperature) and the condenser surface, respectively.

Furthermore, Sherwood number could be evaluated using Equation 5.

$$Sh = \frac{\dot{m}'' D}{\rho D_{aw} (\omega_\infty - \omega_s)} \quad (\text{Eq.5})$$

Where D_{aw} is the binary mass diffusivity of water vapor into air, ρ is the average mixture density, and ω_∞ and ω_s are the vapor mass fractions at the incoming humid air and the condenser surface, respectively. The mass fraction is calculated knowing that the incoming humid air is saturated with water vapor and the condensate at the condenser surface is at the same temperature of the condenser surface. Equation 6 is used to calculate the vapor mass fraction at both conditions.

$$\omega_i = \frac{18 P_{sat}(T_i)}{18P_{sat}(T_i) + 29(P - P_{sat}(T_i))} \quad (\text{Eq.6})$$

Where $P_{sat}(T_i)$ is the saturated pressure at the corresponding temperature T_i , and P is the system pressure.

The experimental study focuses on specific parameters, while numerical analysis using ANSYS will explore a broader range, including target areas, H/D ratios, velocity, and jet spacing (S/D).

3.2.1. UNCERTAINTY ANALYSIS

Uncertainty analysis was conducted to assess the reliability of the experimental results in this study. The main sources of uncertainty included instrumentation errors (thermometers, thermocouples, lab scales, and flow meters), calibration errors, measurement errors (water level), and environmental factors such as ambient temperature and relative humidity. The uncertainties were calculated using standard propagation methods as shown in Equation (7):

$$U_y = \sqrt{\left(\frac{\partial y}{\partial x}\right)^2 U_x^2} \quad (\text{Eq.7})$$

where U is the variable uncertainty, y represents the calculated value, and x are the independent variables. Based on the obtained or estimated precision and accuracy of the measuring instruments used in the study, the uncertainty values for the recorded parameters were determined. For example, the uncertainty for temperature measurement ($\pm 0.1^\circ\text{C}$) and condensation rate ($\pm 0.01 \text{ g/min}$) resulted in relative uncertainties of 0.29% and 1.15%, respectively, while airflow rate ($\pm 0.2 \text{ cfm}$) had a relative uncertainty of 8%. Table 8

summarizes the key parameters and their associated uncertainties, ensuring robust conclusions drawn within the quantified bounds.

Table 8 The uncertainty approximation values

Parameter	Uncertainty ($\pm$)	Relative Uncertainty (%)
Temperature ($^{\circ}\text{C}$)	0.1	0.29%
Condensation Rate (g/min)	0.01	1.15%
Airflow Rate (cfm)	0.2	8%

3.3. NUMERICAL METHODOLOGY

3.3.1. Problem Statement

ANSYS Fluent was employed in this research for analyzing multiphase fluid flow, as the simulations involved long iteration times and extensive computations. The numerical investigation focused on optimizing a 25-jets plate arranged in an inline array by examining various critical parameters, such as the perforated plate-to-condensed plate distance, nozzle spacing, inlet jet temperature, inlet velocity, and target surface area. Each parameter was individually adjusted to determine its best value, with all other parameters held constant. This step-by-step approach enabled a detailed analysis of the parameter space to identify the most effective combination for enhancing heat and mass transfer efficiency. To support this, a 3D computational domain was generated using ANSYS Design Modeler, containing the volume of the object with each cell acting as a control volume for solving discretized conservation equations.

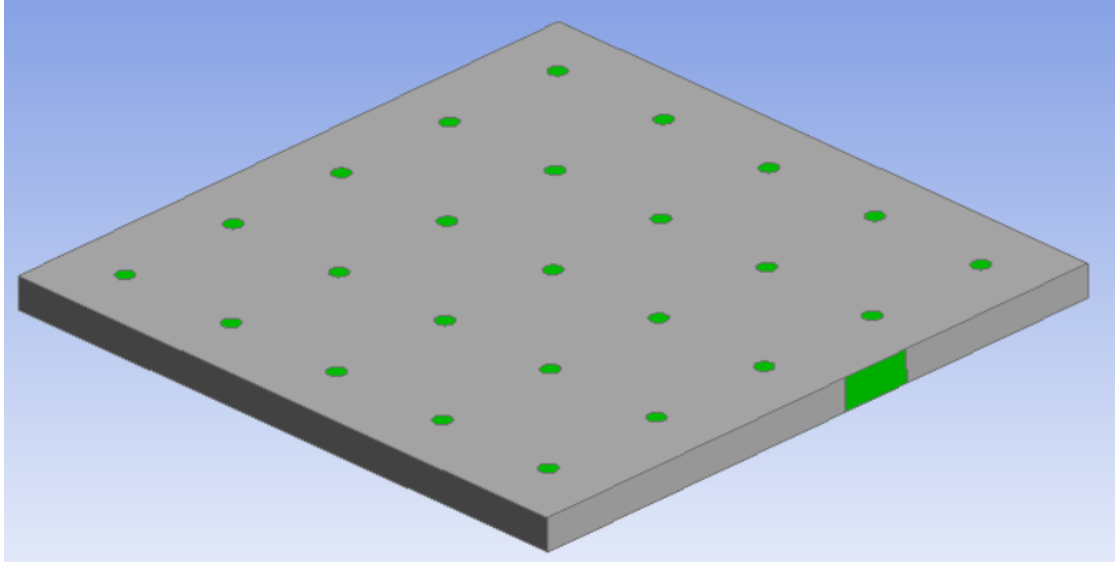


Figure 9 Computational domain.

The domain was prepared by specifying boundary conditions on its surfaces, and the mesh was created using the ANSYS meshing tool. Name selections were defined to assign boundary conditions for the inlet, outlet, and condenser plate, enabling accurate and precise simulation of the jet impingement dynamics. This numerical investigation complemented the experimental methodology, facilitating the exploration of a broader range of conditions and contributing to the development of energy-efficient and cost-effective jet impingement condensers.

3.3.2. Boundary Conditions

Boundary conditions are essential for ensuring the accuracy and reliability of the numerical simulations in this study. The condensed plate is set to a constant temperature of 278 K with a mass fraction of 0.0054. The mixture properties used in the simulations include a density of 0.789 kg/m³, thermal conductivity of 0.0261 W/m·K, viscosity of 0.0000134 Pa·s, Prandtl number of 0.95, mass diffusivity of 0.000023 m²/s, and latent heat of 2384.27 kJ/kg. These values are critical for accurately modeling heat and mass transfer.

The study examines several parameters, such as the perforated plate-to-condensed plate distance (H/D : 2–20), nozzle spacing (S/D : 2–18), inlet jet temperature (331.7–340.8 K), inlet velocity (5–90 m/s), and target area (1225–5625 mm²). In each simulation, one parameter is varied to find its optimal value, while the other parameters are kept constant. Once the best value is identified, it is fixed, and another parameter is adjusted. This step-by-step method ensures a thorough exploration of the parameter space to find the best combination for improving heat and mass transfer efficiency.

The boundary conditions are applied by setting the condensed plate with fixed temperature and mass fraction values. The inlet is defined with varying temperature and velocity ranges, while the outlet allows free outflow. The target area and geometry of the perforated plate are adjusted based on the parameter being tested. Simulations are conducted using pressure-based models under steady-state conditions, including gravity at 9.81 m/s². These settings ensure that fluid flow and heat transfer are modeled accurately, capturing the key interactions that drive efficient heat and mass transfer.

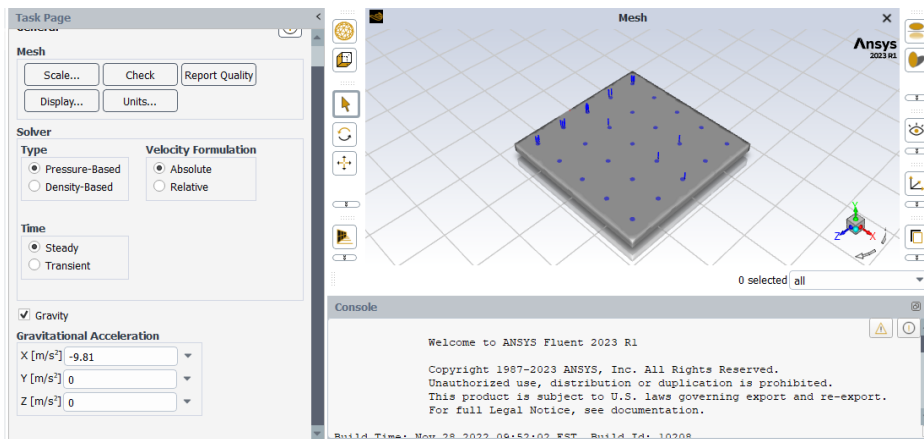


Figure 10 Boundary condition.

3.3.3. Governing Equations for CFD Simulation

The governing equations are used to mathematically describe the physics of fluid flow. The continuity equation, momentum equation, are also known as Navier stroke equations, these equations are needed to describe the state of fluid flow and are solved for all types of CFD modelling. The equations used as following (C) continuity, (D) momentum, (E) energy, (F) species transport and (G) turbulence modeling equations.

C. Continuity Equation

The continuity equation is a fundamental conservation law that expresses the principle of mass conservation. It ensures that mass is neither created nor destroyed within the system. In the context of jet impingement condensation, the continuity equation is used to track the flow of the fluid (usually water vapor in condensation simulations) and ensures that the mass of vapor entering the system is conserved. This is especially important in accurately predicting the velocity and flow distribution at the jet nozzle and the condenser plate.

$$\nabla \cdot \rho \vec{v} = 0 \quad (\text{Eq.8})$$

Where $\vec{v}$ is the velocity vector field.

In jet impingement, the flow is generally incompressible, so the continuity equation ensures that the velocity field remains divergence-free, which is necessary for correct mass flow calculation.

D. Momentum Equation

The momentum equations, as a fundamental component of the Navier-Stokes equations, emphasize the balance of forces acting on a fluid element, including pressure, viscous, and

external forces. These equations describe how the fluid moves under these influences, making them critical for modelling velocity and pressure field variations in jet impingement systems. In such systems, they play a pivotal role in predicting how the jet interacts with the condenser plate, including the evolution of velocity gradients and shear stress near the surface, which are essential for accurately simulating heat and mass transfer during condensation.

Momentum equation (simplified form):

$$\frac{\partial(\rho\vec{v})}{\partial t} + \nabla \cdot (\rho\vec{v} \vec{v}) = -\nabla p + \nabla \cdot (\mu(\nabla\vec{v} + \nabla\vec{v}^T)) + \vec{f} \quad (\text{Eq.9})$$

Where $\vec{v}$ is the field velocity, ρ is the density, p is the pressure, ν is the kinematic viscosity and $\vec{f}$ is external force such as gravity.

The momentum equation accounts for the acceleration of fluid particles due to both internal (pressure and viscous forces) and external forces (gravity or surface interaction). The accurate solution of this equation is key to capturing the correct jet dynamics and interaction with the surface, which influences the condensation rate.

E. Energy Equation

In addition to the fluid mechanics equations, the energy equation is essential for modelling the thermal behavior of the fluid and surface. This equation accounts for the heat transfer due to conduction, convection, and phase change (condensation in this case). The temperature distribution in the fluid and on the surface is crucial for predicting the condensation rate and efficiency of water production.

$$\frac{\partial H}{\partial t} + \nabla \cdot (\vec{v} H) = \nabla \cdot (k\nabla T) + Q \quad (\text{Eq.10})$$

Where T is temperature, H is the total enthalpy of the mixture, k is thermal conductivity, and Q is the internal heat generation.

This equation allows you to model the temperature changes within the fluid as it interacts with the colder condenser surface, driving the condensation process.

F. Species Transport Equation:

When species transport is turned on in ANSYS, the following sets of equations come into play: Species Transport Equation (for each species): The species transport equation describes the conservation of mass for each species in the mixture, considering diffusion, convection, and sources or sinks (such as phase change in condensation).

The general form of the species transport equations for species.

$$\frac{\partial(\rho Y_i)}{\partial t} + \nabla \cdot (\rho \vec{v} Y_i) = \nabla \cdot (D_{ij} \nabla Y_i) + \dot{\omega}_i \quad (\text{Eq.11})$$

Where Y_i is the mass fraction of species, D_{ij} is the molecular diffusivity of species and $\dot{\omega}_i$ is the rate of generation or consumption of species i .

Turning on species transport in ANSYS enables the simulation of multiple fluid species and their interactions within the flow domain. This capability is crucial for modeling processes like jet impingement condensation, where water vapor condenses on a surface, and for scenarios involving gas or vapor mixing with air or other gases. By activating species transport, ANSYS solves for the mass fraction of each species in the flow, providing insight into their behavior and impact on heat and mass transfer, as well as phase changes.

In condensation problems, species transport facilitates the modeling of water vapor transitioning into liquid water on a surface, such as a condenser plate. This process involves capturing the latent heat released during the phase change. The species transport equation

tracks the mass fraction of water vapor (or other species) as it condenses or evaporates, ensuring accurate representation of interactions and transformations in the system.

G. Turbulence Modeling:

The $k - \epsilon$ model predicts well far from the boundaries, $k - \omega$ model predicts well near to the wall, sst model is combination of k-omega and k-epsilon model. The $k - \omega$ model of turbulence is capable of solving the turbulence parameter very close to the boundary or wall region. To capture the phenomenon in such region, fine mesh near to the wall is required.

The governing equations for the $k - \omega$ model:

Equation of Turbulent Kinetic Energy (k):

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k v_i) = \frac{\partial}{\partial x_i}\left(\Gamma_k \frac{\partial k}{\partial x_i}\right) + \tilde{G}_k - Y_k + S_k \quad (\text{Eq.12})$$

Where k is the turbulent kinetic energy, v_i is the velocity component in the i direction, Γ_k is the effective diffusivity, $\tilde{G}_k$ is the generation of turbulent kinetic energy due to mean velocity gradients, Y_k is the dissipation of turbulent kinetic energy and S_k is the Source term.

Equation of Specific Dissipation Rate (ω):

$$\frac{\partial}{\partial t}(\rho \omega) + \frac{\partial}{\partial x_i}(\rho \omega v_i) = \frac{\partial}{\partial x_i}\left(\Gamma_\omega \frac{\partial \omega}{\partial x_i}\right) + G_\omega - Y_\omega + D_\omega + S_\omega \quad (\text{Eq.13})$$

Where Γ_ω is the effective diffusivity of ω , G_ω is the generation of ω due to mean velocity gradients, D_ω is the cross-diffusion term, Y_ω is the dissipation of ω .

The $k - \omega$ model is widely used due to its capability to accurately resolve near-wall turbulence, making it a preferred choice in applications involving boundary-layer flows.

Its reliance on fine mesh ensures enhanced resolution of flow properties close to walls, critical for high-fidelity simulations.

In simulation of jet impingement condensation, the role of following governing equation is crucial. Continuity and Navier-Stokes's equations work together to ensure that the fluid flow is physically accurate, considering mass conservation, viscous effects, and external forces like gravity or buoyancy. Momentum equations help to define the velocity field of the fluid under the action of pressures and viscous forces, which are critical for understanding how the jet impacts the surface. Energy equation is helpful to simulate how heat is transferred from the fluid to the surface and vice versa, modelling the heat exchange necessary for condensation to occur efficiently.

By solving these governing equations simultaneously in ANSYS, one can accurately model the complex interactions of fluid dynamics, heat transfer, and condensation, which are crucial for optimizing water production efficiency in jet impingement condensation systems.

3.3.4. Computational power

CFD simulation requires high-performance computing for efficiency, necessitating workstations with fast CPUs and multiple cores. An HP workstation with an Intel Xeon W-1290 processor (10 cores, 3.2 GHz base speed, up to 5.2 GHz) was used to accelerate Ansys performance and reduce computational time. The workstation features Intel UHD Graphics P630 to minimize rendering time during post-processing and 32GB DDR4 ECC RAM to enhance reliability and prevent file corruption. Sufficient memory ensures efficient simulation operations by avoiding reliance on slower storage drives. GPU acceleration was also employed to reduce analysis times and improve cost-effectiveness.

3.3.5. Mesh Sensitivity Analysis

The 3D domain is meshed using the ANSYS Workbench meshing tool, dividing the geometry into smaller elements (cells) for finite element analysis. High-quality mesh is essential for accurate CFD solutions, as it simplifies the computational domain into manageable elements while maintaining geometric integrity.

H. Mesh Size Variation in ANSYS Meshing Tool

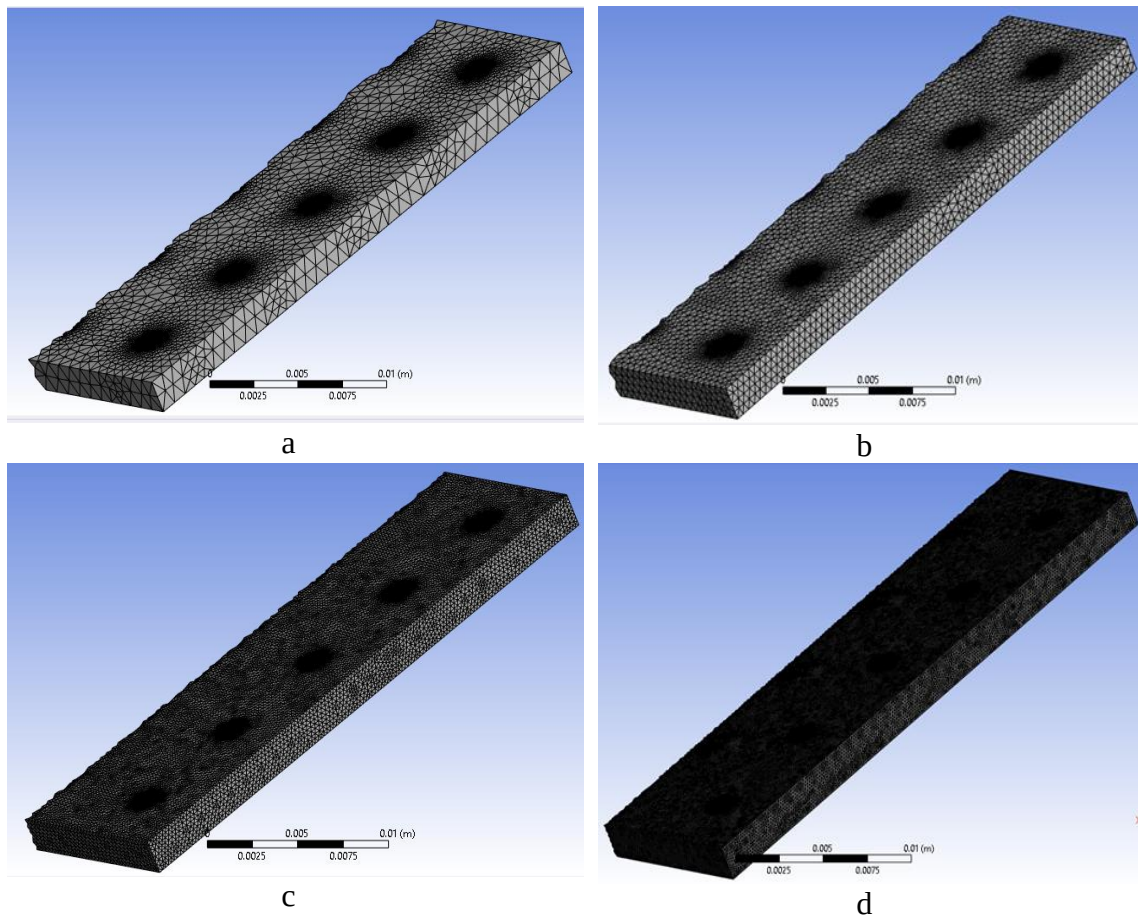


Figure 11 Mesh generation on computational domain with different element sizes (a) 0.001, (b) 0.0005, (c) 0.00025 and (d) 0.00015.

In ANSYS simulations, varying mesh element sizes (0.001, 0.0005, 0.00025, and 0.00015 m) are used to balance accuracy and computational performance. The element size of 0.001 m, which was used in the simulation, showed a minor error of approximately 21.41%

compared to experimental results. This mesh provided accurate results while significantly saving computational time compared to finer mesh sizes, which require more resources.

3.3.6. Thermophysical Properties of Working Fluids

In the simulation of jet impingement condensation, accurate representation of the thermophysical properties of water vapor and air is crucial for precise modeling of heat and mass transfer phenomena. Table 8 summarizes the key properties used in the numerical model, including density, thermal conductivity, viscosity, specific heat, and mass diffusivity.

Table 9 Properties of fluids.

Property name	Water vapor	Air
Density (kg/m^3)	0.804	1.184
Thermal conductivity ($W/m\cdot K$)	0.019	0.0257
Viscosity ($Pa\cdot s$)	9.08×10^{-6}	18.2×10^{-6}
Specific heat ($J/kg\cdot K$)	1.86×10^3	1.005×10^3
Mass diffusivity (m^2/s)	2.6×10^{-5}	

These properties significantly influence the prediction of flow behavior, phase change processes, and energy transfer within the domain.

3.3.7. Simulation Setting

The simulation setting, i.e., automatic time step selection, is carefully done to avoid the problem of divergence in the velocity field. The number of iterations was set to 1000, and the residuals, representing the acceptable error in all governing equations, were maintained

at 10^{-6} . This ensures the accuracy and stability of the numerical solution throughout the simulation process.

3.3.8. Validation of the model.

The numerical model was validated by comparing its predictions with experimental results, as shown in Figure 12. The figure illustrates a close agreement between the experimental data (denoted by markers) and the numerical predictions (represented by a continuous line), with an error margin of approximately 21.41%. This error is considered small and acceptable for the studied system, indicating the reliability of the model in capturing the key physical phenomena.

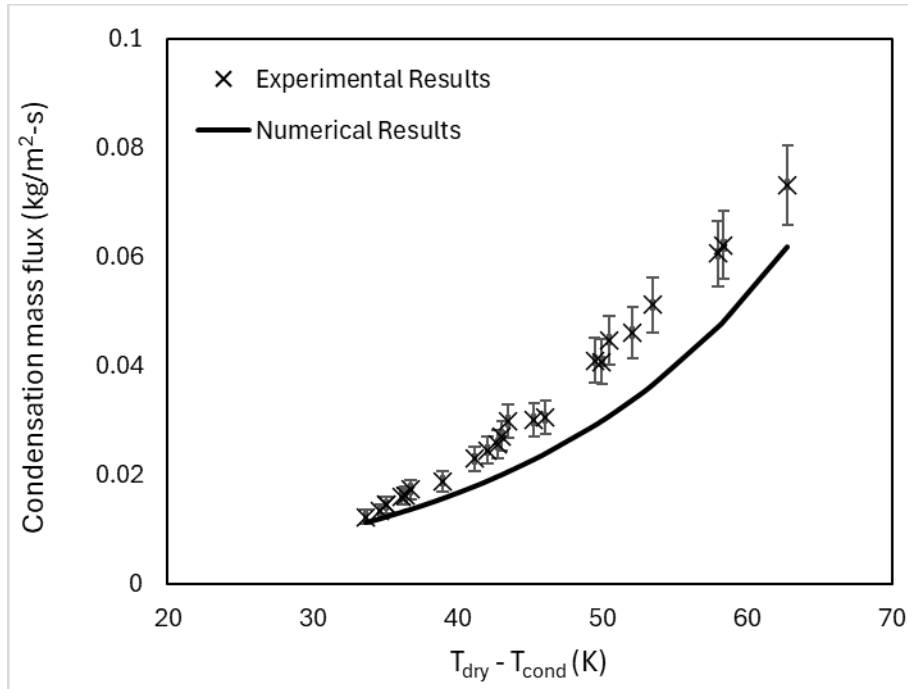


Figure 12 Model validation.

The alignment of the data points and the strong correlation between the two datasets further confirm the accuracy of the model, demonstrating its capability to effectively simulate the system's behavior within the acceptable error range.

CHAPTER 4

RESULTS AND DISCUSSION

4.1. EXPERIMENTAL RESULTS AND DISCUSSION

4.1.1. Bubble Column Analysis

The two bubble columns in the experimental setup play an important role in maintaining controlled conditions at the inlet of the JIC. This ensures fair and consistent comparisons when evaluating different parameters of the JIC. Humidity levels were measured using wet-bulb and dry-bulb temperatures, with the relative humidity at the exit of the second column always exceeding 99%. Figure 13 shows how the humid air temperature exiting the second column relates to the temperature of the second heater, with different settings for the first heater. The temperature of heater 2, shown on the x-axis, has a clear and consistent effect on T_{dry} . As the temperature of heater 2 increases, T_{dry} rises steadily, regardless of the temperature of heater 1. For example, when the temperature of heater 2 reaches 75°C, T_{dry} approaches 66.5°C.

While increasing heater 1's temperature does slightly raise the humid air temperature, its impact is much smaller. For instance, when heater 2 is set to 40°C, raising heater 1 from 45°C to 80°C increases the humid air temperature from 37.5°C to 44°C. In contrast, keeping heater 1 at 80°C and increasing heater 2's temperature from 40°C to 75°C results in a much larger jump in humid air temperature from 44°C to 67.8°C.

When both heaters are set to high values (around 75°C or higher), T_{dry} reaches its peak, close to 66-70°C. Interestingly, you don't need to maximize both heaters to achieve similar results. For example, a T_{dry} of 62-65°C can be achieved with heater 1 set to 70-75°C and heater 2 set to 65-70°C.

This shows that heater 2 plays the leading role in controlling T_{dry} , while heater 1 acts as a secondary source of heat. By focusing on heater 2, it's possible to get the desired results without needing to push both heaters to their limits. Understanding this relationship is key to creating the right thermal conditions without using more energy than necessary.

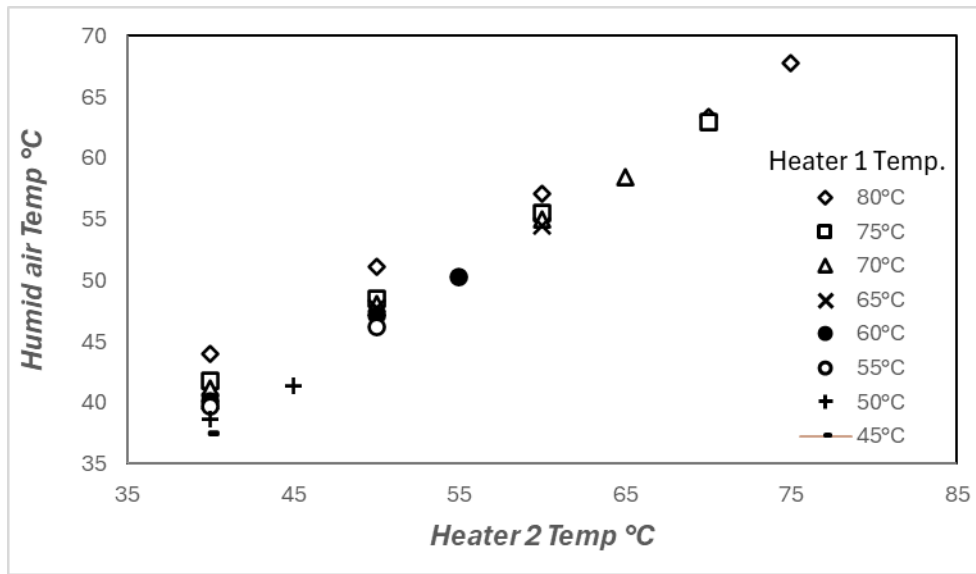


Figure 13 The relationship between the heater temperatures and T_{dry} .

4.1.2. Nusselt Number Analysis

Figures 14(a-c) illustrate the relationship between the Nusselt number (Nu) and the Jakob number (Ja) for the three configurations investigated in the study. The x-axis represents the Jakob number (Ja), a dimensionless parameter characterizing the ratio of sensible to latent heat, while the y-axis shows the corresponding Nusselt number (Nu), a dimensionless parameter indicating the intensity of convective heat transfer.

Across all configurations, Configuration 1 (Figure 14a) demonstrates the highest Nusselt number values, indicating superior heat transfer performance. The relationship for this configuration follows the equation,

$$Nu = 3029(Ja)^{1.8543} \quad (\text{Eq.14})$$

This strong performance is attributed to the smaller jet-to-plate distance and tighter nozzle spacing, which significantly enhances cooling effectiveness by increasing turbulence and heat transfer rates. Configuration 2 (Figure 14b) shows a moderate reduction in performance, with the Nusselt number following:

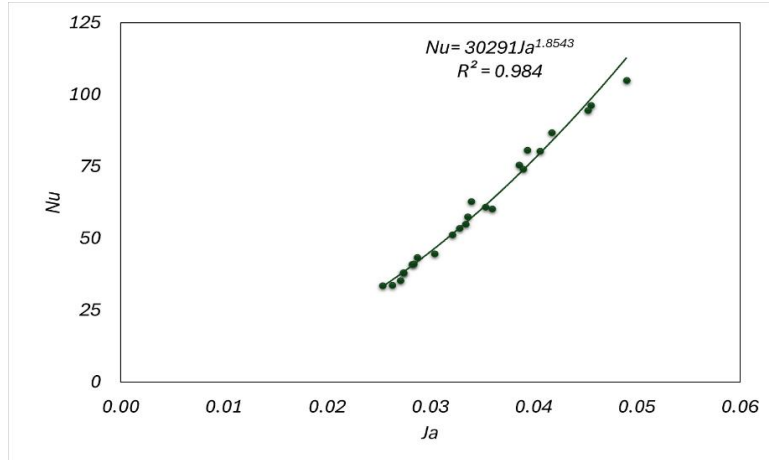
$$Nu = 2472(Ja)^{1.375} \quad (\text{Eq.15})$$

The slightly larger plate area and wider nozzle spacing in this configuration reduce the convective intensity, lowering the overall heat transfer rate. Lastly, Configuration 3 (Figure 14c) exhibits the lowest Nusselt number values, described by the equation:

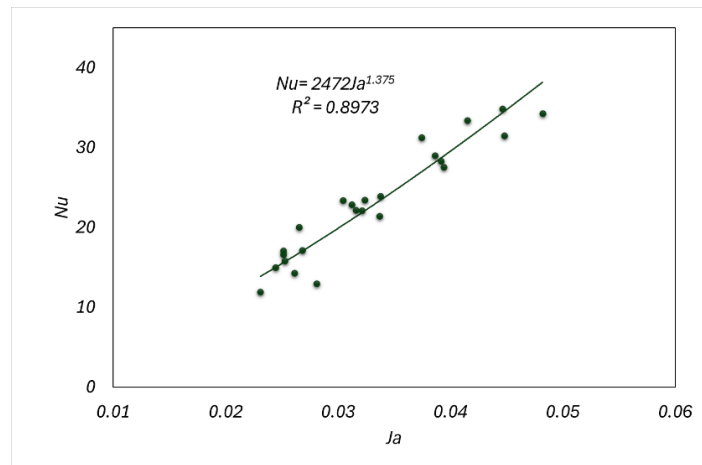
$$Nu = 436.05(Ja)^{0.8253} \quad (\text{Eq.16})$$

This reduction in heat transfer performance is attributed to the significantly larger jet-to-plate distance, which decreases the impact of velocity and turbulence intensity, thereby diminishing cooling effectiveness.

(14a)



(14b)



(14c)

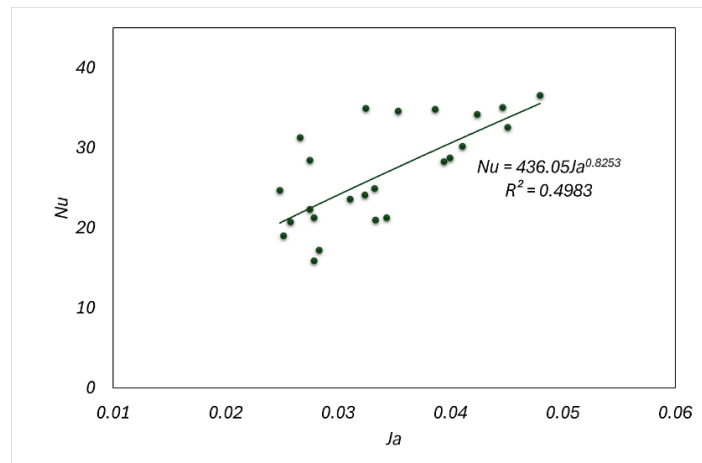


Figure 14.(a-c) Nu vs Ja of all configurations respectively.

4.1.3. Sherwood Number Analysis

Figure 15(a-c) illustrates the relationship between the Sherwood number (Sh) and the Jakob number (Ja) for the three configurations examined in the study. The x-axis represents the Jakob number (Ja), a dimensionless parameter that indicates the ratio of sensible to latent heat, while the y-axis shows the Sherwood number (Sh), a dimensionless parameter representing mass transfer performance.

Among the configurations, Configuration 1 (Figure 15a) exhibits the highest Sherwood number, described by the equation

$$Sh = 1E + 7(Ja)^{5.4519} \quad (\text{Eq.17})$$

This strong performance can be attributed to the focused jet impingement, facilitated by the smallest jet-to-plate distance, which enhances mass transfer intensity. Configuration 2 (Figure 15b) shows a slight reduction in Sherwood number, following

$$Sh = 2E + 6(Ja)^{5.0575} \quad (\text{Eq.18})$$

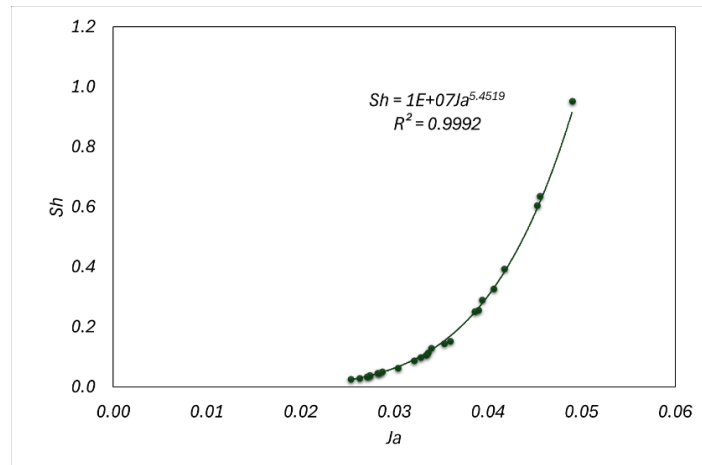
The larger plate area and wider nozzle spacing in this configuration moderately lower the mass transfer intensity. Lastly, Configuration 3 (Figure 15c) demonstrates the lowest Sherwood number, represented by

$$Sh = 271674(Ja)^{4.561} \quad (\text{Eq.19})$$

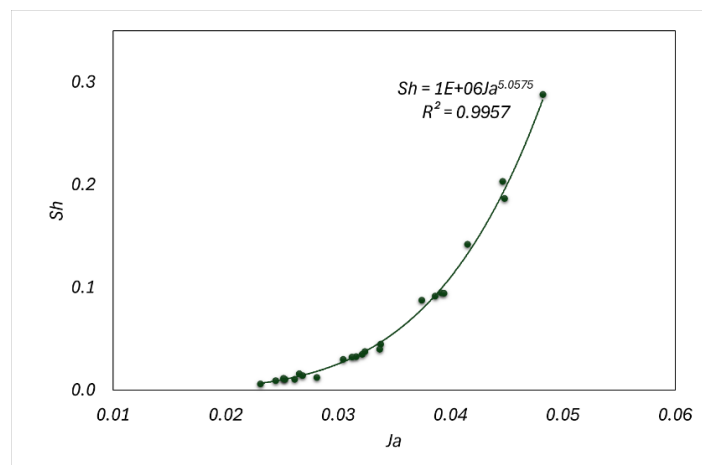
The larger jet-to-plate distance in this configuration reduces the impact strength of the jets, leading to reduced condensation and mass transfer rates. These results underscore that the first configuration achieves the best mass transfer due to optimized geometric and

operational parameters, while the third configuration's larger spacing leads to the least effective mass transfer.

(15a)



(15b)



(15c)

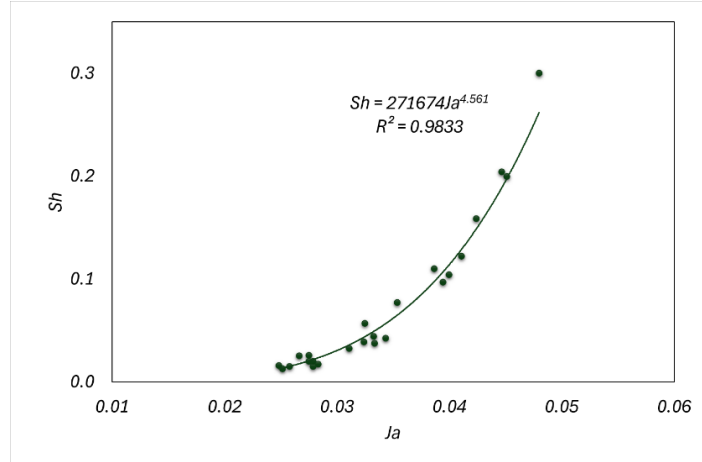


Figure 15 (a-c) Sh vs Ja of all configurations respectively.

The Sherwood number data aligns with the trends observed for heat transfer, demonstrating that the smaller jet-to-plate distance in the first configuration leads to better mass transfer, while the third configuration, with the largest jet-to-plate distance, is the least efficient.

4.1.4. Comparative Analysis of Configurations

I. Heat Transfer

Configuration 1 consistently achieves the highest Nu, indicating that the smaller jet-to-plate distance ($H/D = 2$) and tighter nozzle spacing (7 mm) lead to the most effective convective heat transfer. Configuration 2 performs moderately well but shows reduced heat transfer performance due to its larger target plate area and wider nozzle spacing. Configuration 3 exhibits the lowest heat transfer capability, with the increased jet-to-plate distance weakening the cooling intensity.

J. Mass Transfer

Similarly, Configuration 1 also achieves the highest Sh, indicating the most effective mass transfer. The smaller jet-to-plate distance and tighter nozzle spacing promote better

condensation. Configuration 2 shows slightly lower mass transfer performance due to its larger plate area and wider nozzle spacing. Configuration 3 demonstrates the least effective condensation due to the larger jet-to-plate distance, which weakens the jets' impact on the surface.

K. Sensitivity to Operating Conditions

Configuration 3 demonstrates the highest sensitivity to variations in operating conditions, as reflected by the lower R^2 value and greater scatter, especially in the Nusselt number plot. The larger jet-to-plate distance in this configuration reduces the consistency of the jets' impact, making the system more likely to experience fluctuations in performance with changes in temperature difference. In contrast, Configurations 1 and 2 exhibit greater stability under varying conditions, with Configuration 1 showing the most robust behavior due to its optimized jet-to-plate distance and nozzle spacing. These features help maintain consistent heat and mass transfer performance, even under varying operating conditions.

4.2. NUMERICAL RESULTS AND DISCUSSION

The requirement of computers for performing the iteration, importance of mesh, computational time required, development of computational domain, material selection, importance of adjustable and fixed time step, initialization and post processing is discussed in detail before proceeding for the CFD simulation.

4.2.1. Nusselt Number Analysis

L. Effect of Temperature on Nusselt Number

Figure 16 illustrates the relationship between temperature and the Nusselt number, showing a general upward trend. As the temperature increases from approximately 310 K

to 340 K, the Nusselt number consistently rises, reaching its highest value at the maximum temperature. This trend aligns with the principle that higher temperature gradients enhance convective heat transfer, as measured by the Nusselt number.

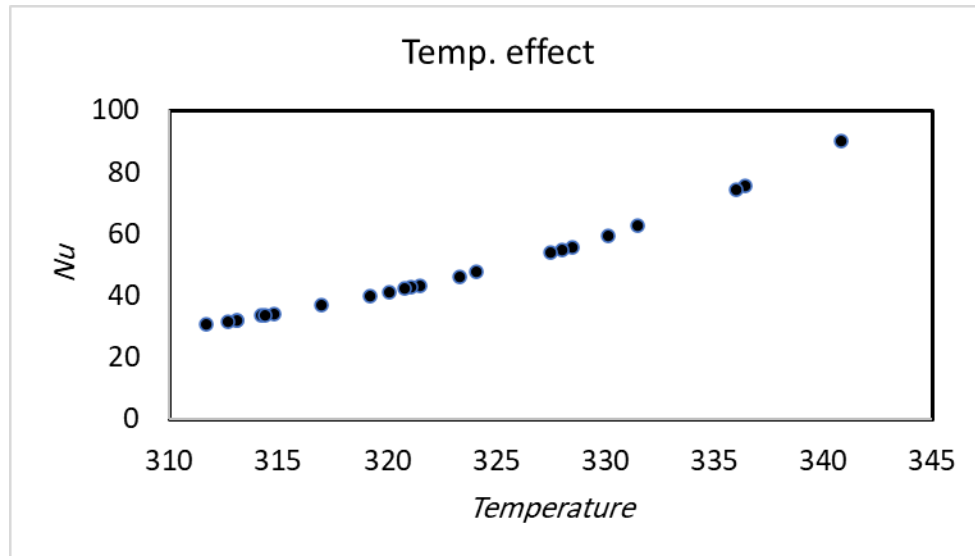


Figure 16 Nusselt Number Vs Temperature.

At lower temperatures, the thermal driving force is reduced, which explains the lower Nusselt numbers observed in this range. As the temperature increases, the convective heat transfer becomes more efficient due to a stronger thermal gradient, resulting in higher Nusselt numbers. This effect is particularly evident in the smooth rise of the data points as the temperature approaches the upper limit.

While there may be minor fluctuations in the Nusselt number at intermediate temperatures, these are likely due to secondary factors such as flow dynamics, fluid properties, or measurement uncertainties. However, the overall trend clearly indicates that temperature plays a dominant role in dictating the effectiveness of convective heat transfer.

In conclusion, the figure demonstrates a strong positive correlation between temperature and the Nusselt number, emphasizing that higher temperatures lead to more effective convective heat transfer. This behavior is consistent with the theoretical understanding of heat transfer, where greater temperature differences drive more efficient convection.

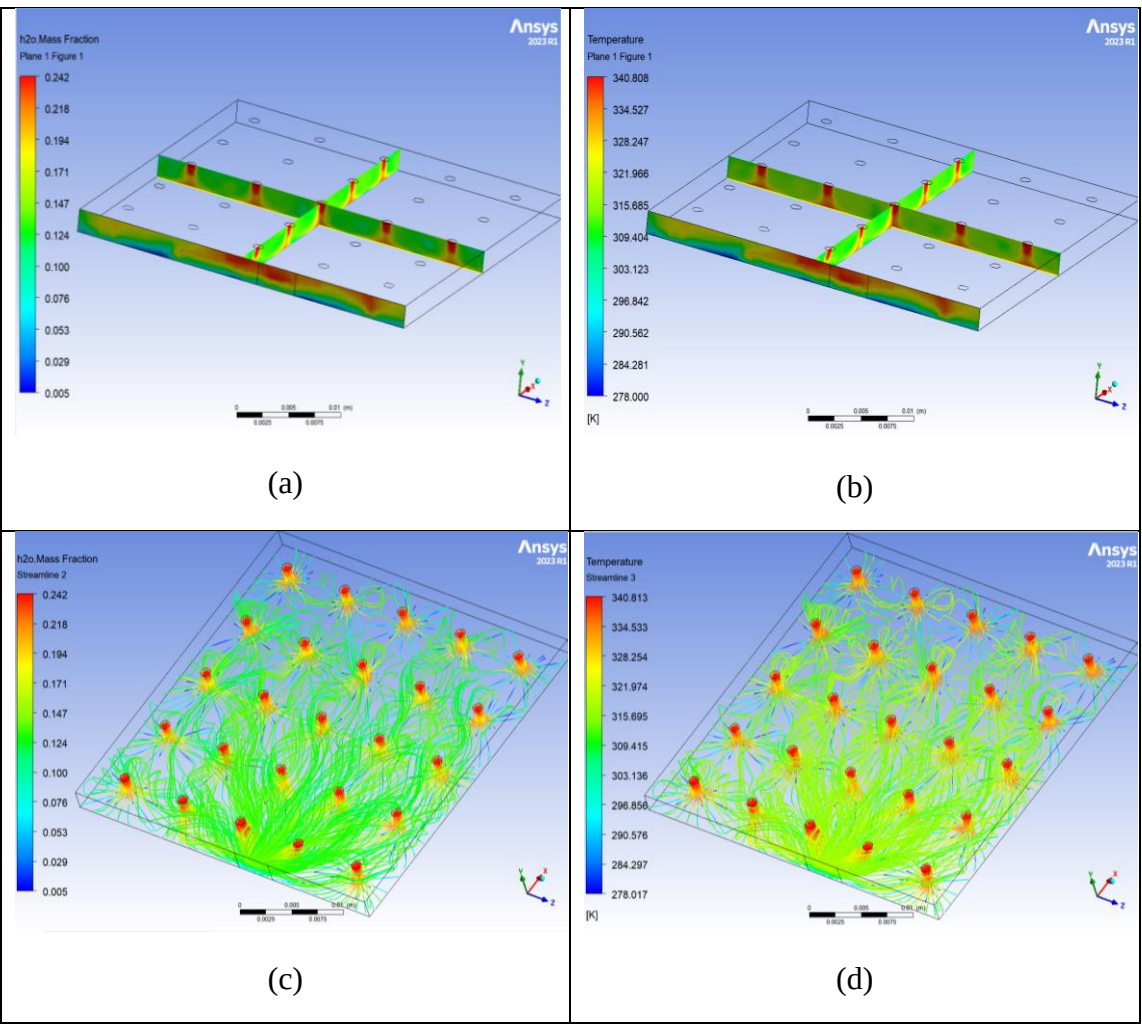


Figure 17 Contours of (a) H₂O Mass Fraction and (b) Temperature Distribution on Plane for Jet Impingement Flow at varying temperature test of inlet vapor. Streamline Visualization of (c) H₂O and (d) Temperature Mass Fraction Distribution in Jet Impingement Configuration at various temperature of inlet vapor

Building upon the analysis of the temperature and Nusselt number relationship, Figure 17(a-d) provide further insights into the mass and heat transfer dynamics within the jet impingement system. Figure 17(a) shows the H₂O mass fraction distributions, illustrating how water vapor concentration peaks near jet exits and decreases outward due to mixing

and potential condensation. Figure 17(b) depicts the temperature distribution, highlighting localized heating near the jets that gradually transition to cooler regions as the fluid spreads. Figures 17(c) and 17(d) present streamline visualizations for mass fraction and temperature, respectively, demonstrating complex flow patterns, efficient mixing, cooling, and heat transfer driven by turbulent jet interactions. Figure 17(a-d) prove that temperature gradients and flow dynamics critically influence convective heat transfer efficiency, as reflected by the Nusselt number behavior, further emphasizing the role of optimized jet and system parameters for enhanced heat and mass transfer performance.

M. Effect of condensed plate area on Nusselt Number

The chart in figure 18 demonstrates that as the surface area decreases while other parameters such as (vapor temperature, S/D ratio, H/D ratio, and velocity) remain constant, the Nusselt number increases.

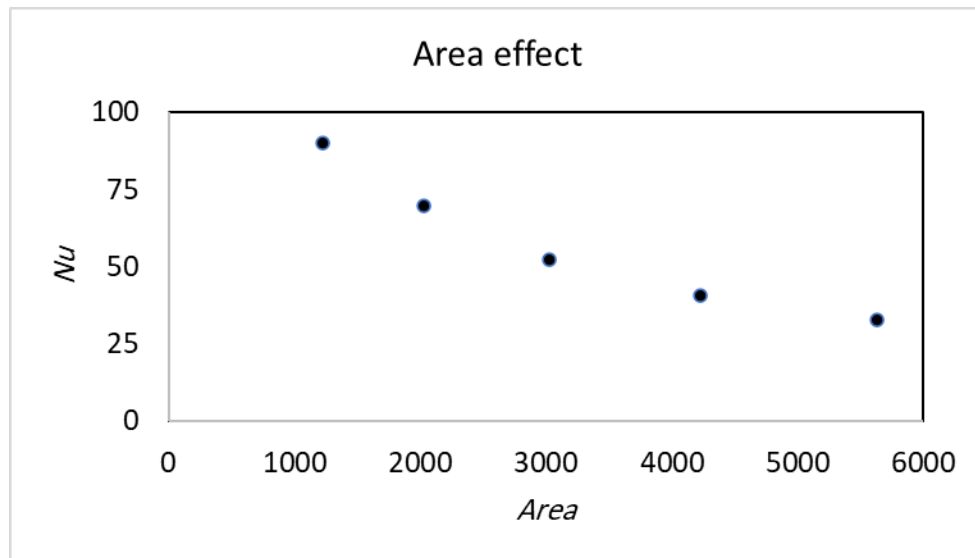


Figure 18 Nusselt Number Vs Surface Area.

Reducing the surface area increases the Nusselt number by concentrating heat transfer, while larger areas enhance total condensation through extended vapor-plate contact. This

balance between localized efficiency and total mass transfer performance is evident in Figures 19(a-d) and 20(a-d), which illustrate heat and mass transfer dynamics for surface areas of 35×35 mm and 75×75 mm, respectively.

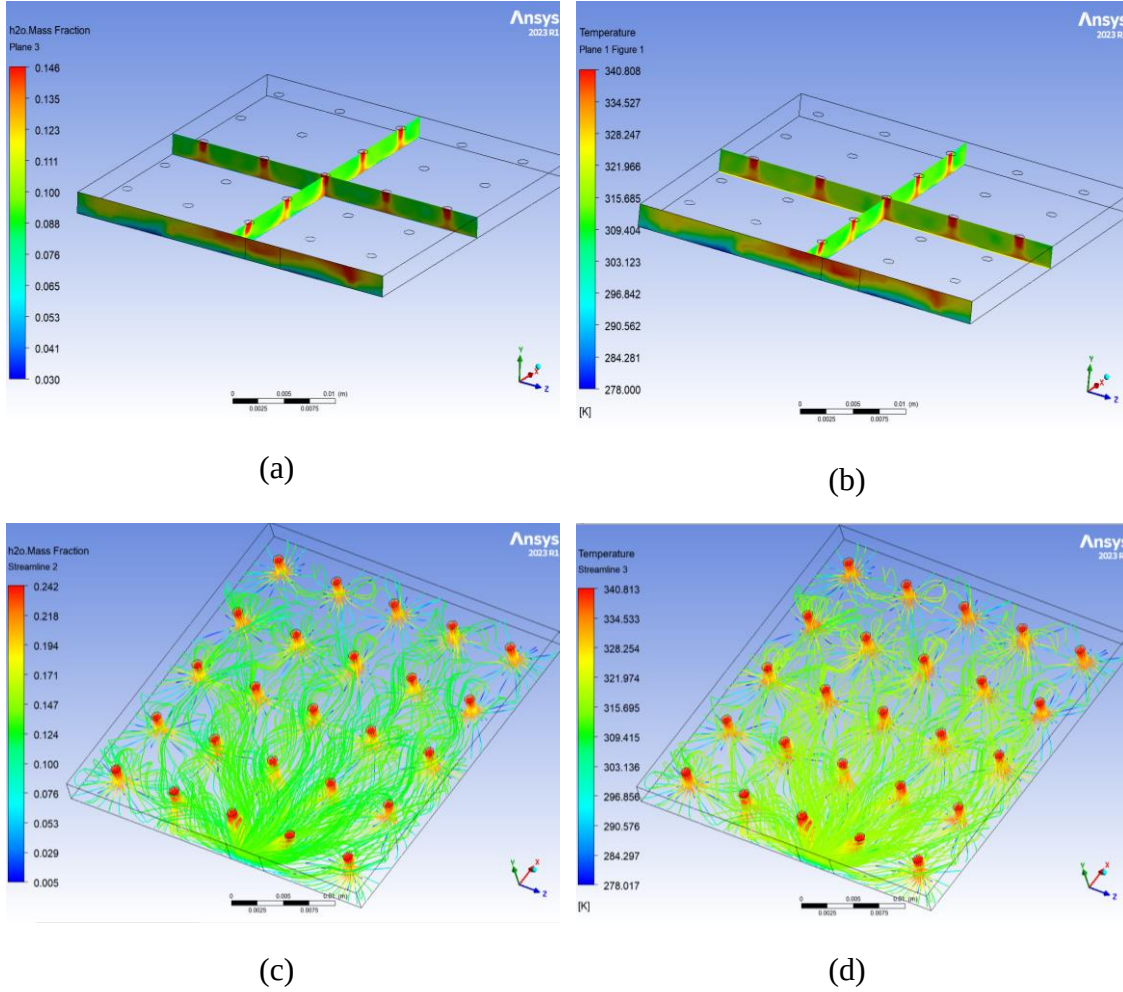


Figure 19 Contours of (a) H₂O Mass Fraction and (b) Temperature Distribution on Plane for Jet Impingement Flow at area 35×35 mm². Streamline Visualization of (c) H₂O and (d) Temperature Mass Fraction Distribution in Jet Impingement Configuration at area 35×35 mm².

For the 35×35 mm² area, Figure 19(a-d) illustrates heat and mass transfer dynamics. High vapor concentration near jet exits (Figure 19a) transitions to lower concentrations as condensation occurs. Temperature distribution (Figure 19b) shows intense heating near the jets with rapid cooling outward, indicating efficient transfer. Streamline visualizations (Figures 19c-d) reveal strong turbulence and effective mixing, with the compact area

enabling concentrated energy transfer, higher Nusselt numbers, and efficient heat and mass transfer.

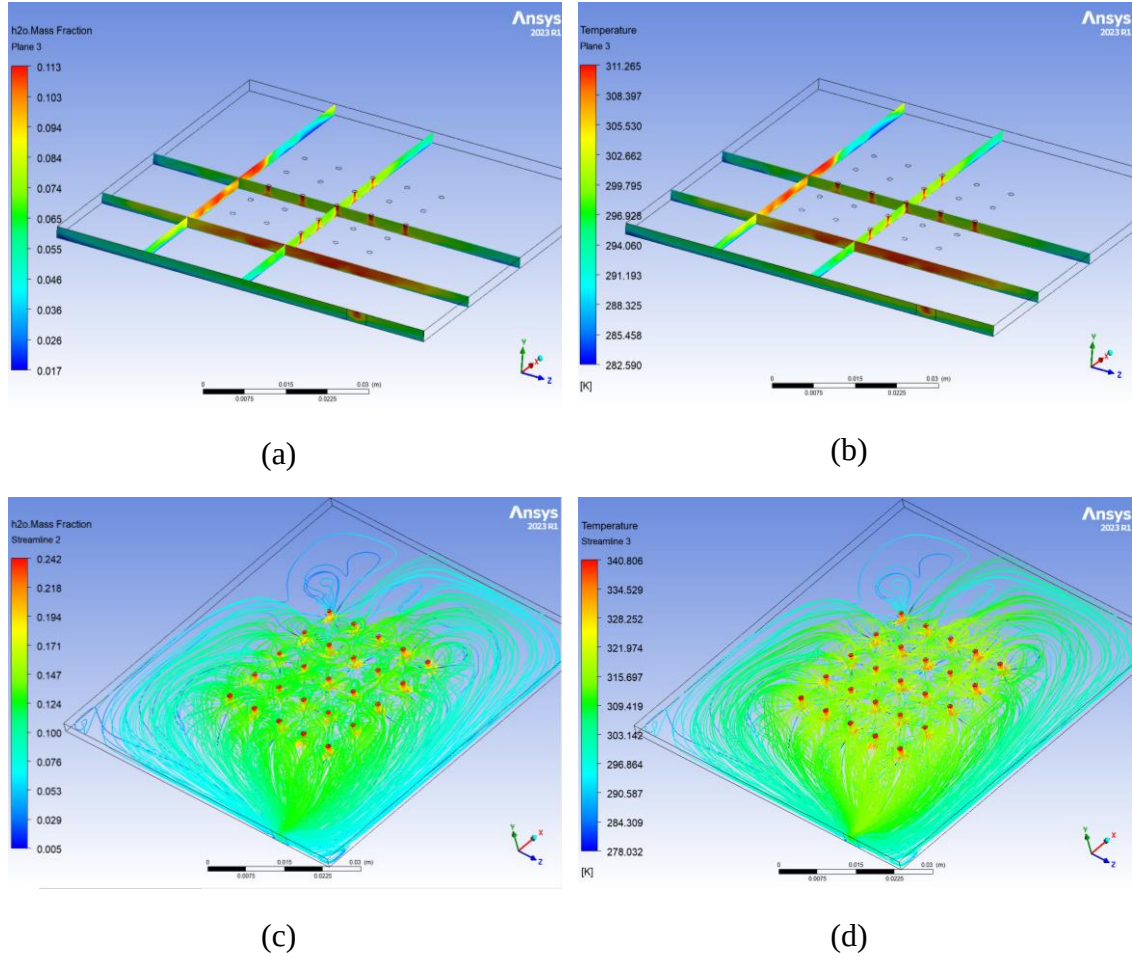


Figure 20 Contours of (a) H₂O Mass Fraction and (b) Temperature Distribution on Plane for Jet Impingement Flow at area 75*75mm². Streamline Visualization of (c) H₂O and (d) Temperature Mass Fraction Distribution in Jet Impingement Configuration at area 75*75mm².

For the 75 × 75 mm² area, Figures 20(a-d) highlight heat and mass transfer dynamics for a larger surface. Figure 20(a) shows high vapor concentrations near jet exits, with less effective spreading and mixing. Figure 20(b) reveals localized cooling near the jets but weaker temperature gradients, reducing heat transfer efficiency. Streamline visualizations (Figures 20c-d) depict broader, less turbulent flows, indicating lower localized heat and mass transfer intensity. While the larger area increases contact surface, it lowers energy

transfer efficiency per unit area, resulting in reduced Nusselt numbers. This illustrates the trade-off between localized efficiency and overall condensation, emphasizing the need for optimization.

N. Effect of S/D Ratio on Nusselt Number

The chart in figure 21 shows the relationship between the Nusselt number and the S/D ratio, with all other parameters fixed. The Nusselt number increases with S/D, peaking at S/D = 15, before slightly declining at higher ratios, reflecting peak convective heat transfer behavior. This highlights the importance of optimizing jet-to-jet spacing for maximum efficiency.

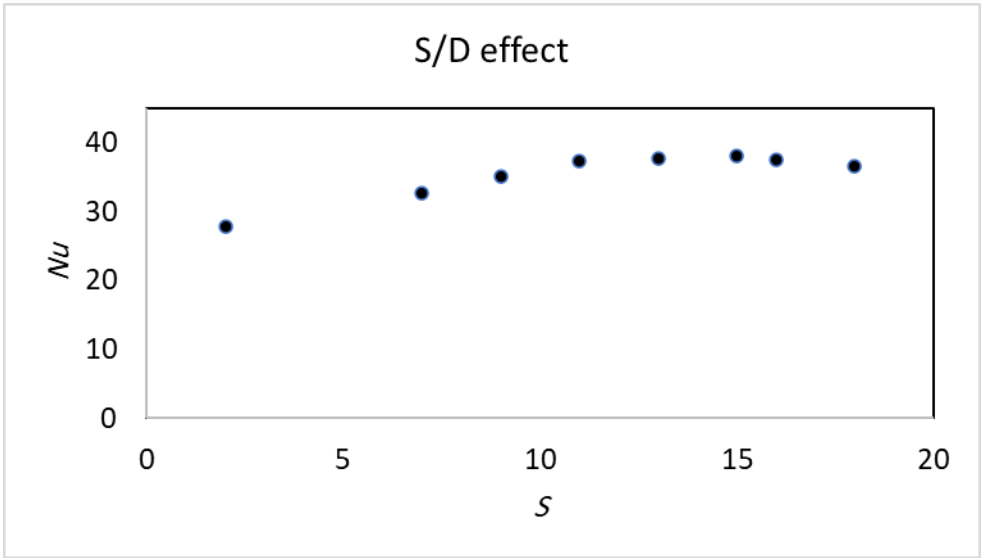


Figure 21 Nusselt Number Vs S/D ratio.

Figures 22(a-d), 23(a-d), and 24(a-d) illustrate flow, temperature, and mass fraction distributions for S/D ratios of 2 mm, 15 mm, and 18 mm, providing detailed insights into the observed trends.

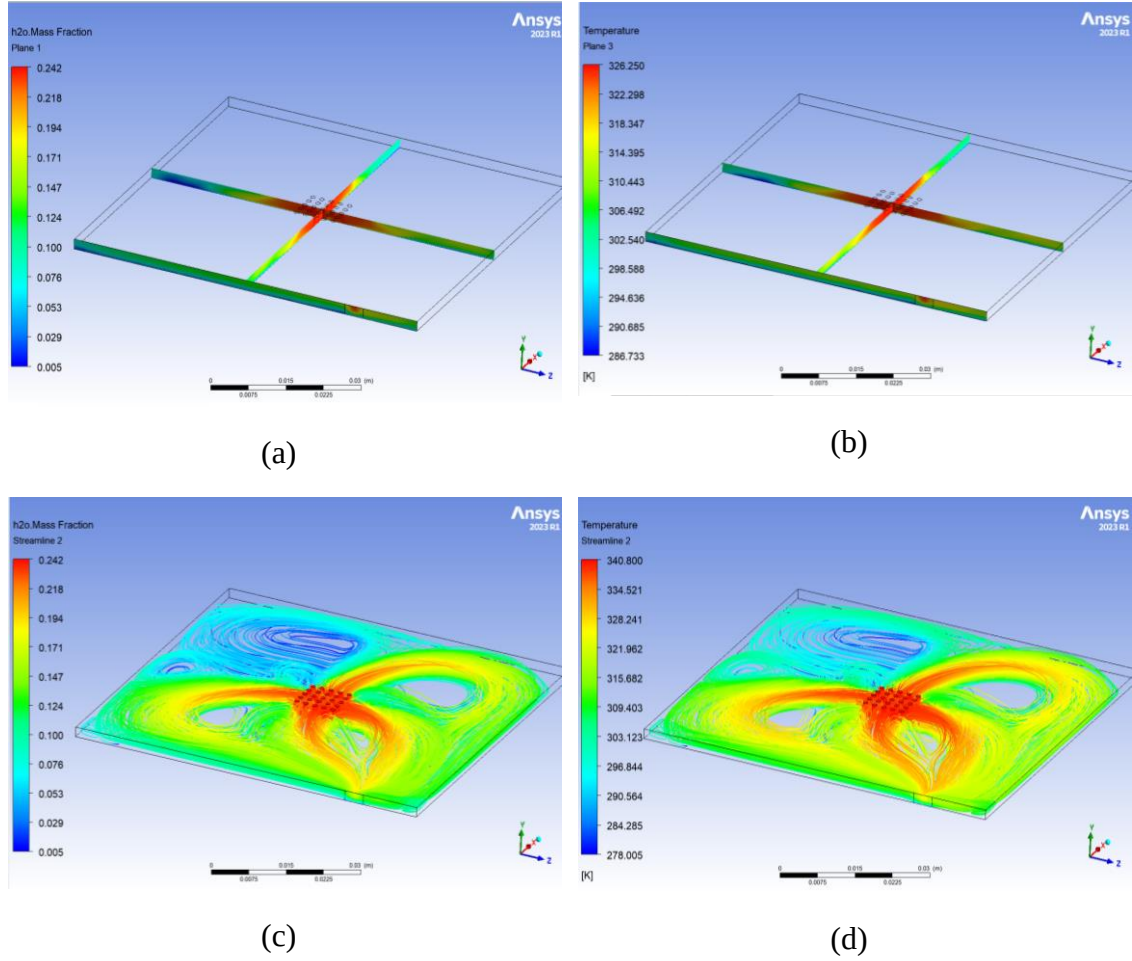


Figure 22 Contours of (a) H₂O Mass Fraction and (b) Temperature Distribution on Plane for Jet Impingement Flow at $S/D=2$. Streamline Visualization of (c) H₂O and (d) Temperature Mass Fraction Distribution in Jet Impingement Configuration at $S/D=2$.

Figures 22(a-d) illustrate the dynamics at a small S/D ratio of 2, where the jets are closely spaced. Figure 22(a) shows the H₂O mass fraction distribution, with limited spreading and overlapping of jet interactions, leading to suboptimal condensation and mixing. Figure 22(b) highlights the temperature distribution, with significant localized heating near the jets but insufficient cooling across the surface. Figures 22(c) and 22(d) depict streamline patterns for mass fraction and temperature, showing concentrated turbulence near the jets but limited interaction between neighboring jets. This closely packed configuration results in lower heat transfer efficiency, corresponding to the lower Nusselt number observed at this S/D ratio.

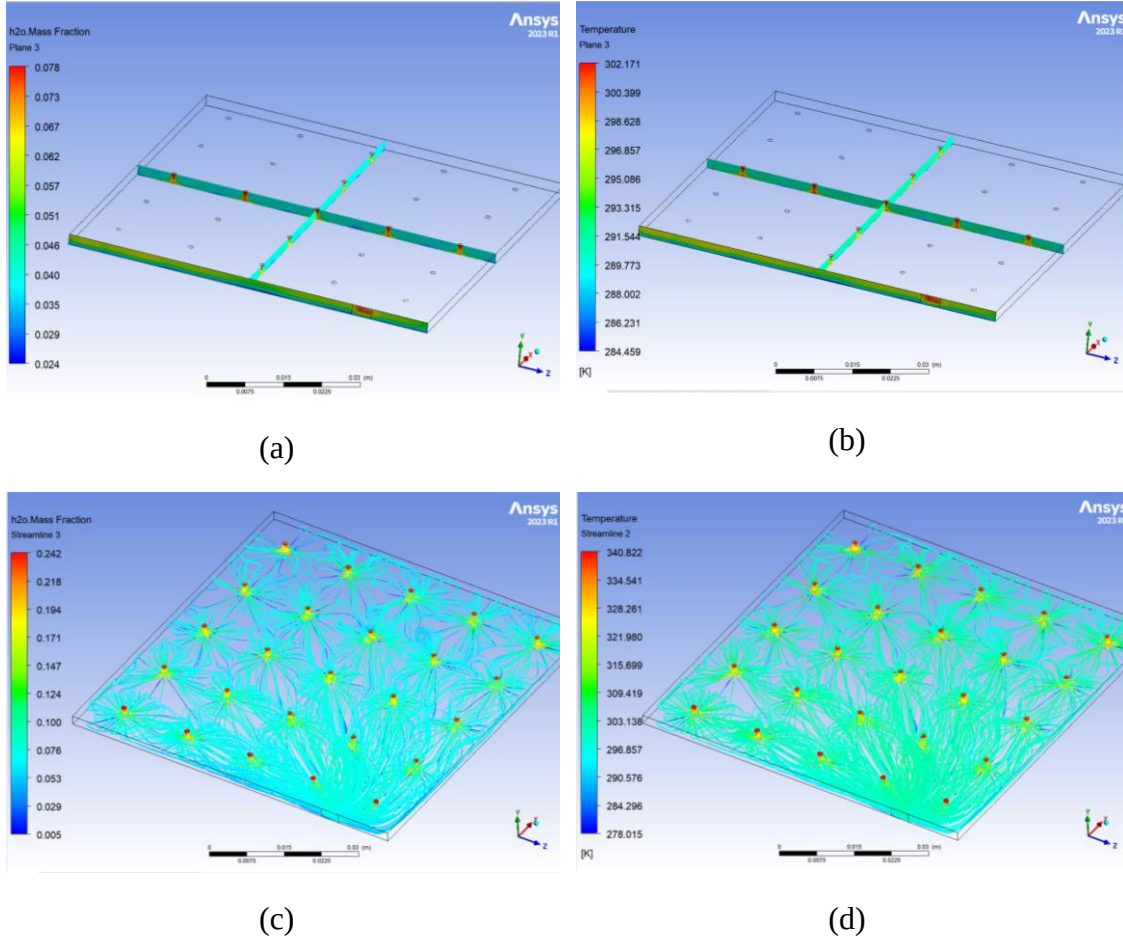


Figure 23 Contours of (a) H₂O Mass Fraction and (b) Temperature Distribution on Plane for Jet Impingement Flow at $S/D=15$. Streamline Visualization of (c) H₂O and (d) Temperature Mass Fraction Distribution in Jet Impingement Configuration at $S/D=15$.

Figures 23(a-d) represent the best S/D ratio of 15, where the Nusselt number peaks. Figure 23(a) shows a well-distributed H₂O mass fraction, indicating effective mixing and interaction of vapor with the surface. Figure 23(b) reveals a balanced temperature gradient, with effective cooling across the plate. Figures 23(c) and 23(d) display streamlined flow patterns with well-developed turbulence and strong jet interactions, maximizing heat and mass transfer. This configuration achieves the highest heat transfer efficiency due to the best balance of jet spacing, turbulence, and thermal boundary layer effects.

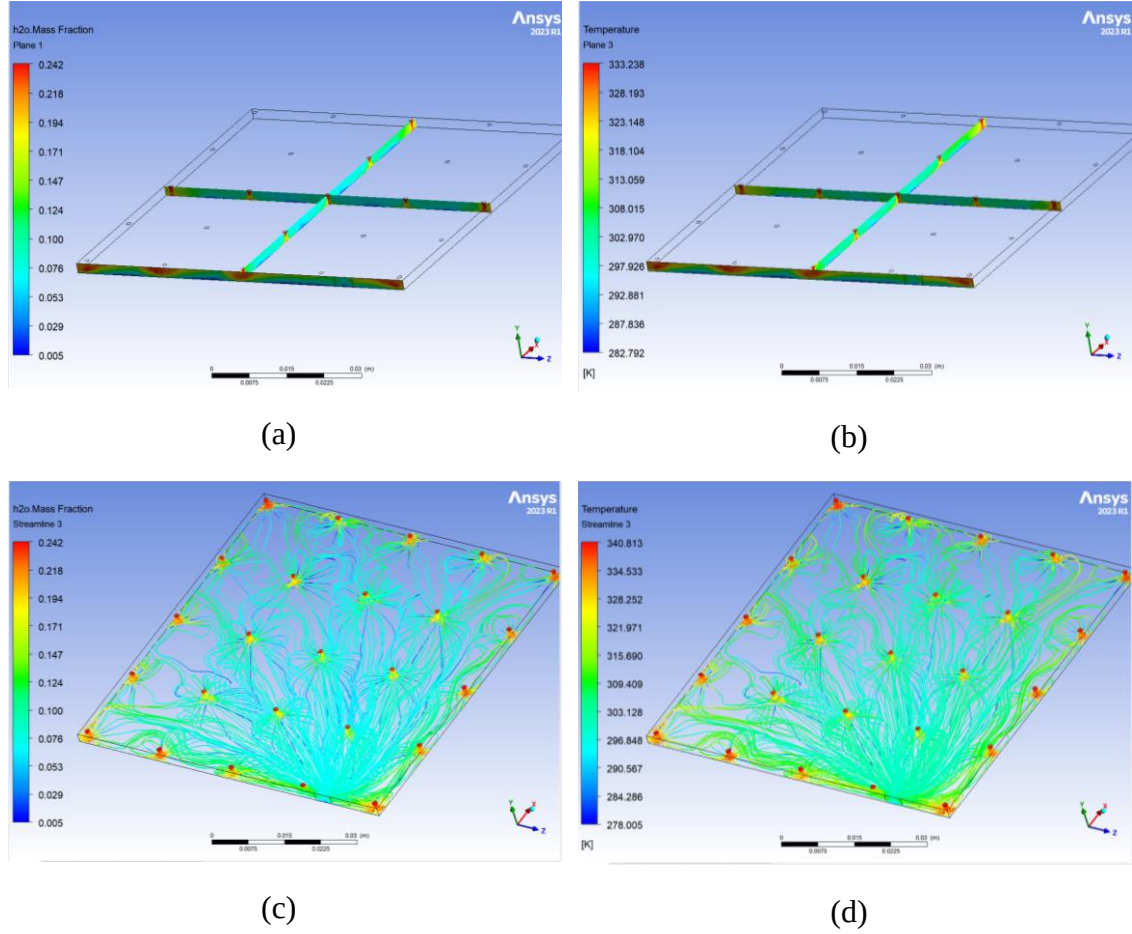


Figure 24 Contours of (a) H₂O Mass Fraction and (b) Temperature Distribution on Plane for Jet Impingement Flow at S/D=18. Streamline Visualization of (c) H₂O and (d) Temperature Mass Fraction Distribution in Jet Impingement Configuration at S/D=18.

Figures 24(a-d) illustrate the performance at an S/D ratio of 18, showing reduced jet interaction and mixing (Figure 24a), weaker temperature gradients (Figure 24b), and diminished turbulence and flow interaction (Figures 24c-d). This leads to less effective heat and mass transfer, consistent with the observed decline in the Nusselt number beyond the best S/D ratio of 15. These visualizations align with Figure 21, emphasizing that while increasing the S/D ratio initially improves heat transfer, excessive spacing weakens turbulence and reduces performance, underscoring the need for optimized jet spacing.

O. Effect of H/D on Nu

Figure 25 shows the Nusselt number variation with the H/D ratio, peaking at $H/D = 2$, where the jet-to-plate distance minimizes, maximizing heat transfer due to strong impingement and turbulence. As H/D increases from 2 to 6, the Nusselt number declines, reflecting weakened jet impact and reduced turbulence. Beyond $H/D = 6$, the Nusselt number stabilizes with minor variations, but after $H/D = 12$, it decreases slightly due to diminished jet interaction and less effective convective heat transfer.

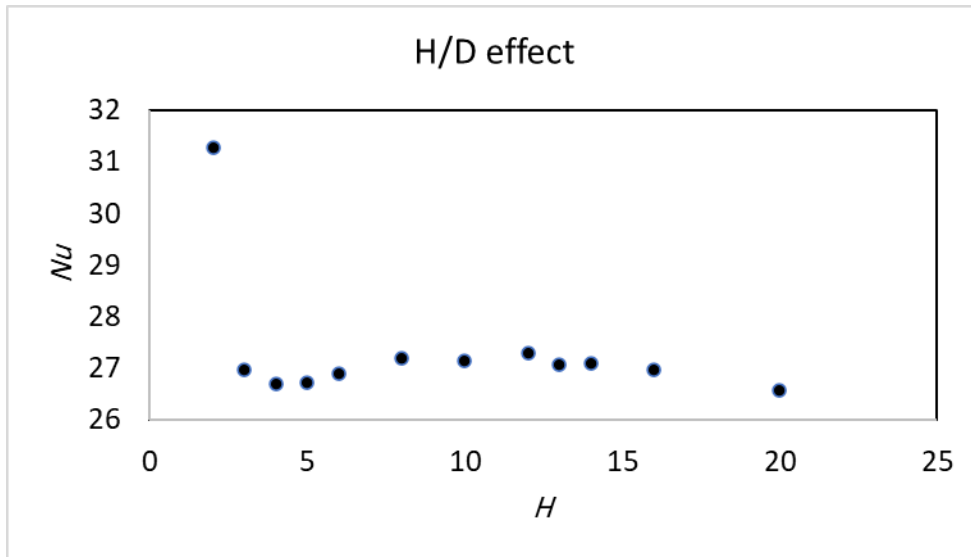


Figure 25 Nusselt Number Vs H/D.

In summary, the highest Nusselt number is observed at $H/D = 2$, after which it declines as the H/D ratio increases. The chart confirms that lower H/D ratios are more favorable for maximizing convective heat transfer efficiency, with the H/D ratio having a non-linear and diminishing effect on performance as it increases. This highlights the importance of maintaining a minimal jet-to-plate distance for best system performance. The following figures 26(a-d) and 27(a-d) provide detailed visualizations of the flow, temperature, and mass fraction distributions for H/D ratios of 2 and 20.

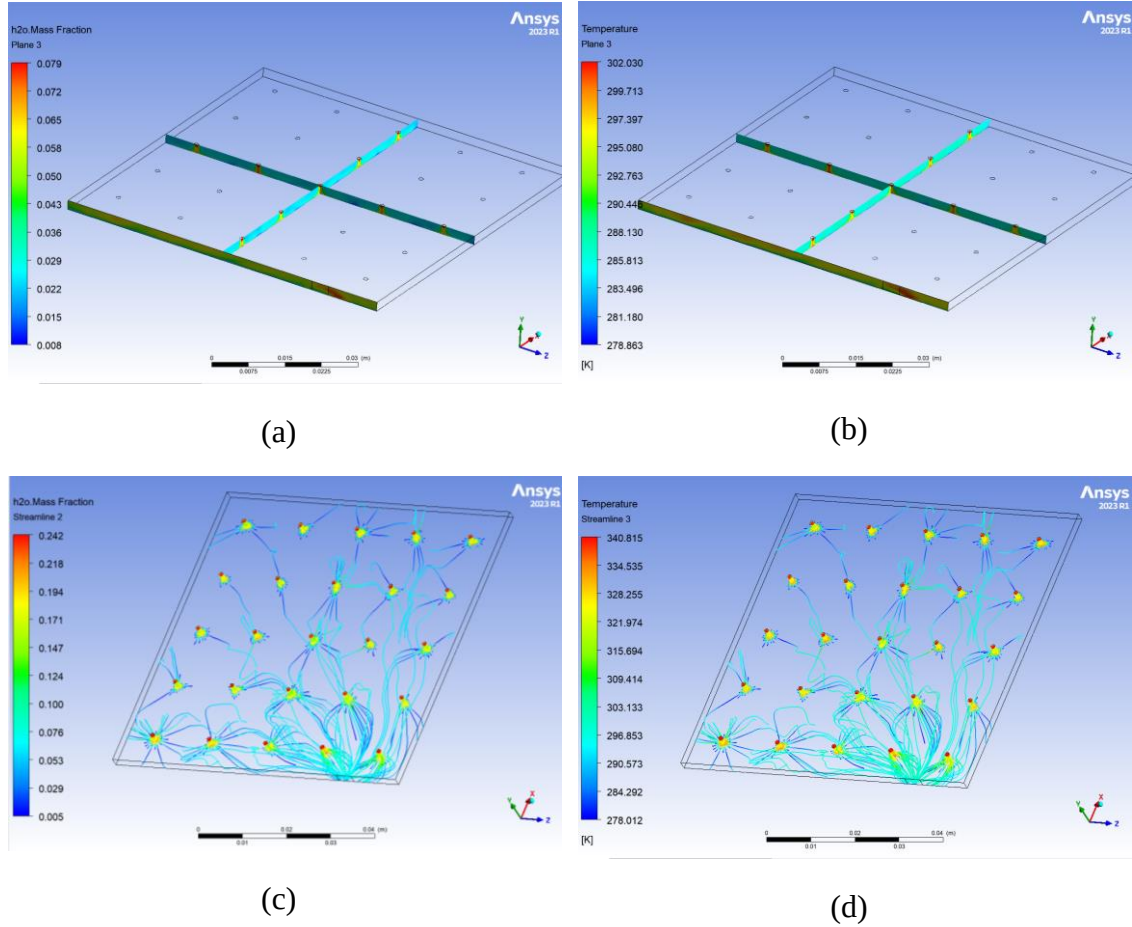


Figure 26 Contours of (a) H₂O Mass Fraction and (b) Temperature Distribution on Plane for Jet Impingement Flow at $H/D=2$. Streamline Visualization of (c) H₂O and (d) Temperature Mass Fraction Distribution in Jet Impingement Configuration at $H/D=2$.

Figures 44(a-d) illustrate the behavior at the smallest H/D ratio of 2. Figure 26(a) shows the H₂O mass fraction distribution, with concentrated interaction and effective mixing near the jet exit points, which enhances condensation. Figure 26(b) displays the temperature distribution, highlighting localized heating around the jet exits and efficient cooling across the surface. Figures 26(c) and 26(d) reveal streamline patterns with intense turbulence and strong jet-to-surface interaction, ensuring maximum convective heat transfer. This configuration corresponds to the peak Nusselt number observed in the chart.

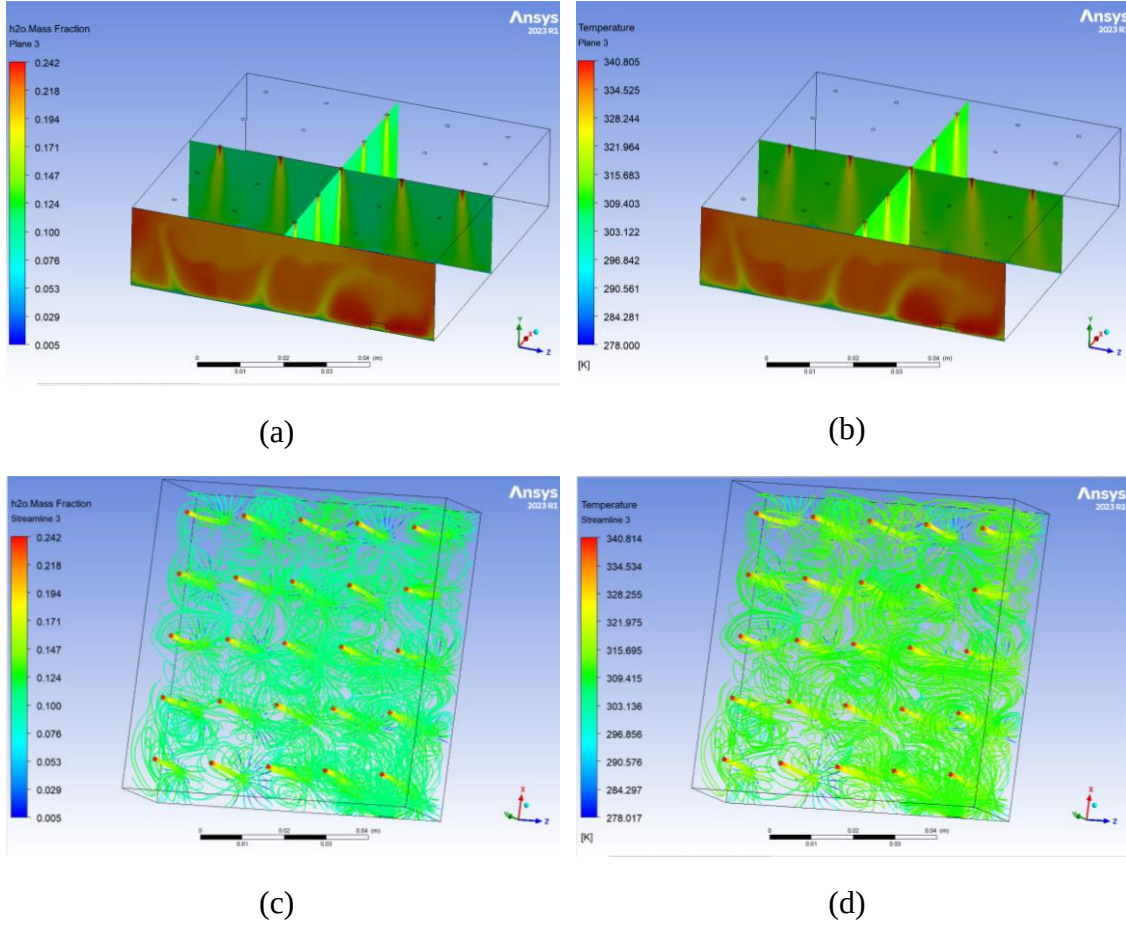


Figure 27 Contours of (a) H₂O Mass Fraction and (b) Temperature Distribution on Plane for Jet Impingement Flow at H/D=20. Streamline Visualization of (c) H₂O and (d) Temperature Mass Fraction Distribution in Jet Impingement Configuration at H/D=20.

Figures 27(a-d) illustrate the dynamics at an H/D ratio of 20, showing limited mixing, reduced condensation efficiency (Figure 27a), weaker temperature gradients (Figure 27b), and diminished turbulence with suboptimal surface interaction (Figures 27c-d). This results in the lowest Nusselt number, aligning with Figure 25 trends. While H/D = 2 maximizes turbulence and heat transfer efficiency, increasing the H/D ratio weakens jet effectiveness, leading to a decline in performance. Maintaining a minimal H/D ratio is essential for optimal thermal performance in jet impingement systems.

P. Effect of Velocity(m/s) on Nusselt Number

Figure 28 shows that as velocity increases from 5 m/s to 90 m/s, the Nusselt number rises from 4.799 to 43.952, driven by enhanced turbulence and fluid mixing. The relationship is non-linear, with the most significant gains at lower velocities. At higher velocities, the flow becomes fully turbulent, reducing the impact of further increases on heat transfer.

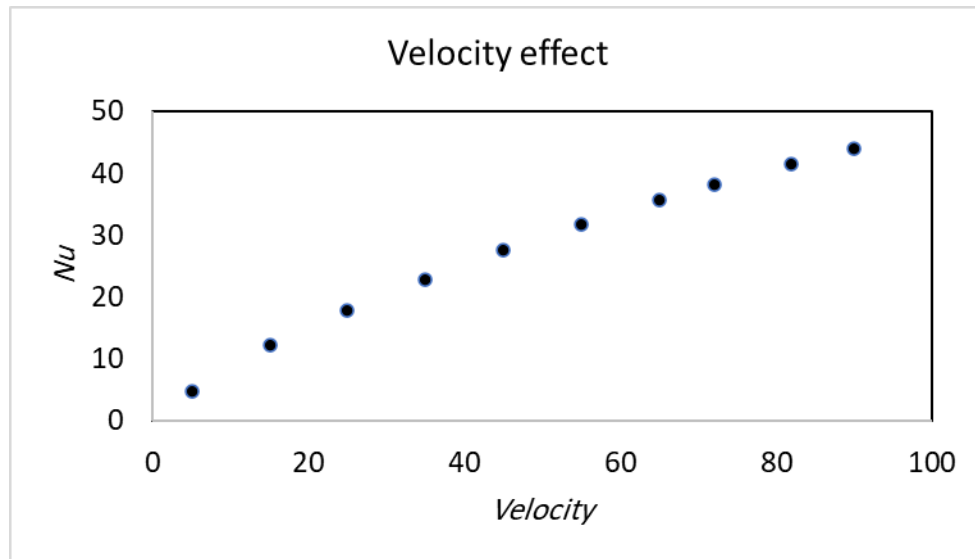


Figure 28 Nusselt Number Vs Velocity(m/s).

At low velocities, the flow tends to be laminar, and heat transfer is primarily dominated by conduction. As velocity increases, the flow transitions to a turbulent state, where convective heat transfer becomes dominant, leading to a rapid increase in the Nusselt number. Once the flow becomes fully turbulent, further increases in velocity have a diminishing impact on the Nusselt number due to the saturation of turbulence effects. Figures 29-31(a-d) provides a deeper visualization of this behavior under varying velocity conditions, demonstrating the progression of flow and thermal profiles across the domain.

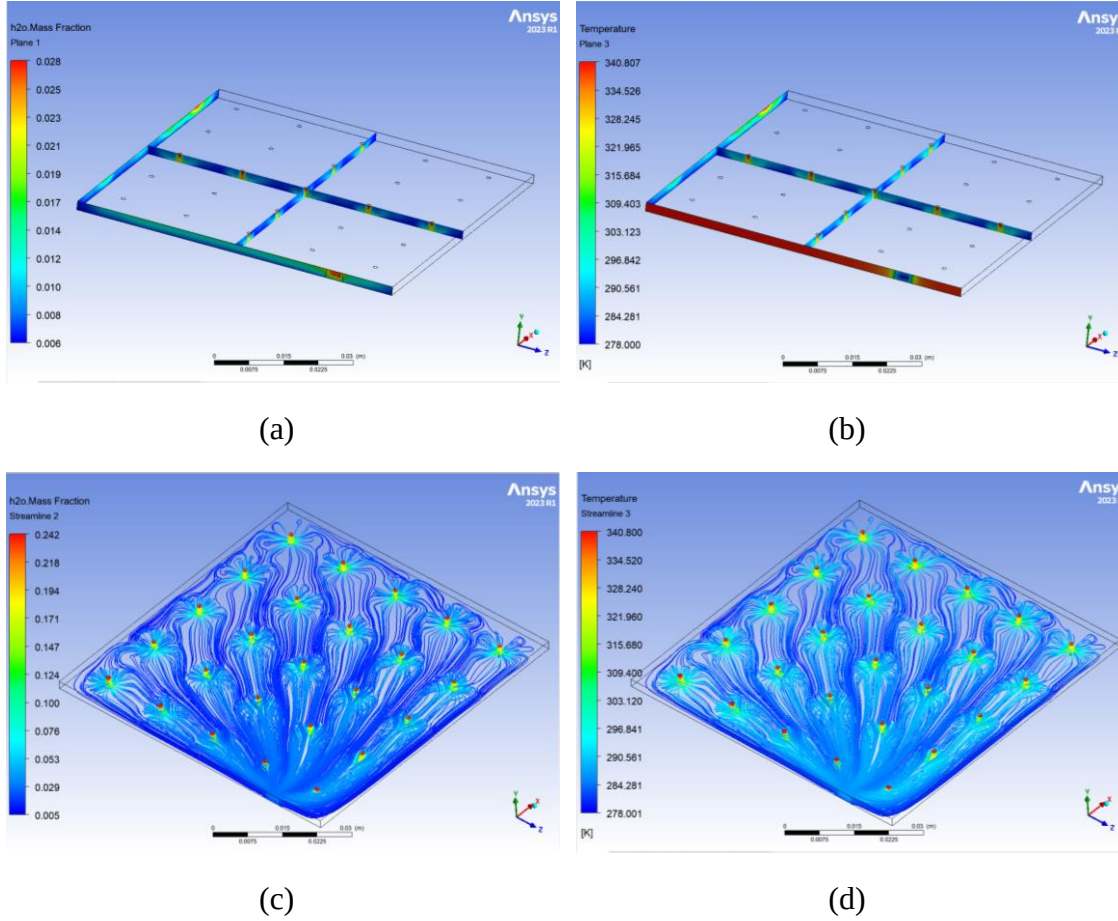


Figure 29 Contours of (a) H₂O Mass Fraction and (b) Temperature Distribution on Plane for Jet Impingement Flow at $v = 5$ m/s. Streamline Visualization of (c) H₂O and (d) Temperature Mass Fraction Distribution in Jet Impingement Configuration at $v = 5$ m/s.

Figures 29(a-d) depict the flow at 5 m/s, where the flow remains predominantly laminar. Figure 29(a) shows a nearly uniform H₂O mass fraction distribution, indicating weak mixing and limited mass transfer. Figure 29(b) reveals a uniform temperature profile with low thermal gradients, suggesting weak heat transfer. Figure 29(c) and (d) show smooth streamlines for both mass fraction and temperature, highlighting the lack of turbulence and cross-flow, resulting in poor mixing and low heat and mass transfer. These observations explain the low Nusselt number, as effective heat transfer relies on turbulence and mixing, which are absent at this velocity.

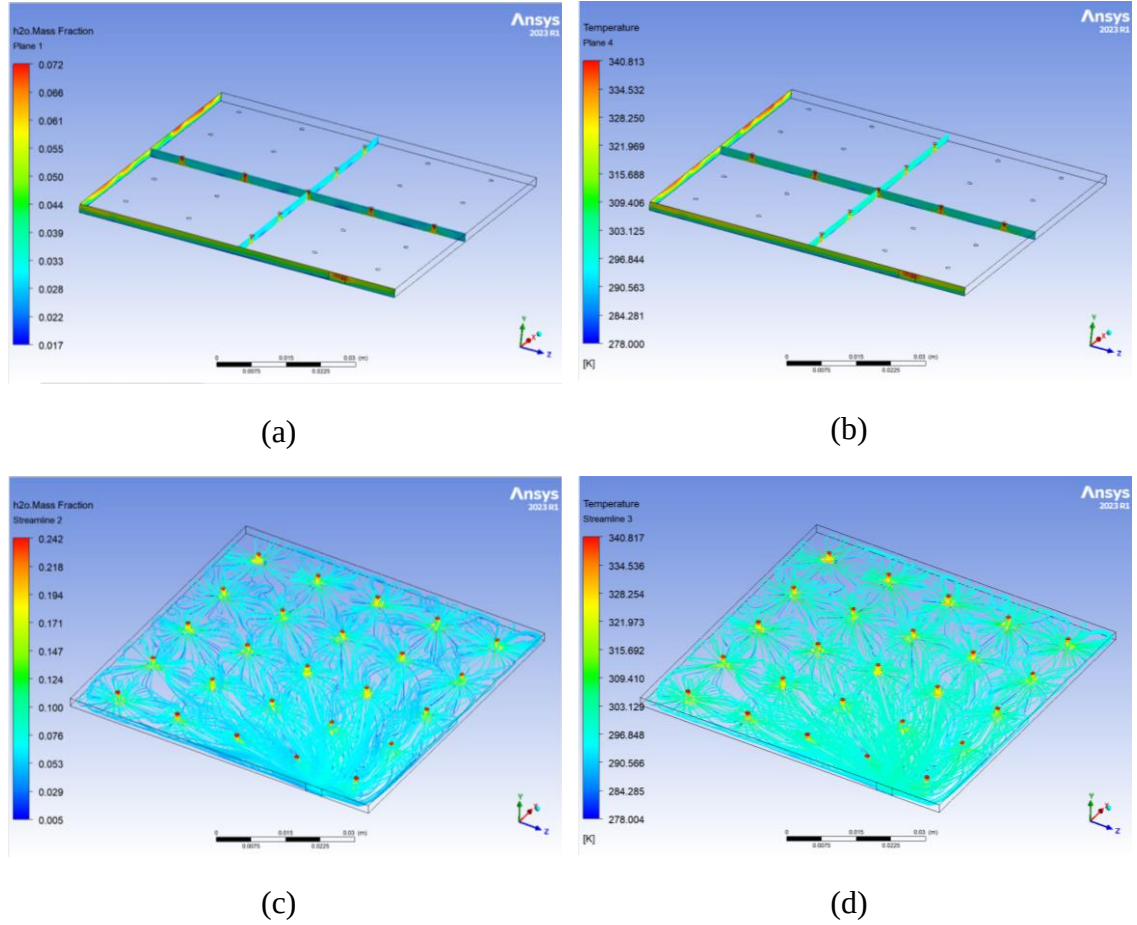


Figure 30 Contours of (a) H₂O Mass Fraction and (b) Temperature Distribution on Plane for Jet Impingement Flow at $v = 45$ m/s. Streamline Visualization of (c) H₂O and (d) Temperature Mass Fraction Distribution in Jet Impingement Configuration at $v = 45$ m/s.

Figures 30(a-d) show the flow at 45 m/s, where turbulence starts to develop. Figure 30(a) shows localized variations in the H₂O mass fraction, indicating enhanced mass transfer. Figure 30(b) reveals steeper temperature gradients, reflecting improved thermal performance. Figures 30(c) and (d) display chaotic flow patterns and increased cross-flow mixing, leading to more uniform moisture distribution and stronger thermal mixing. These observations highlight how higher velocity boosts heat and mass transfer, as seen in the increased Nusselt number at this intermediate velocity.

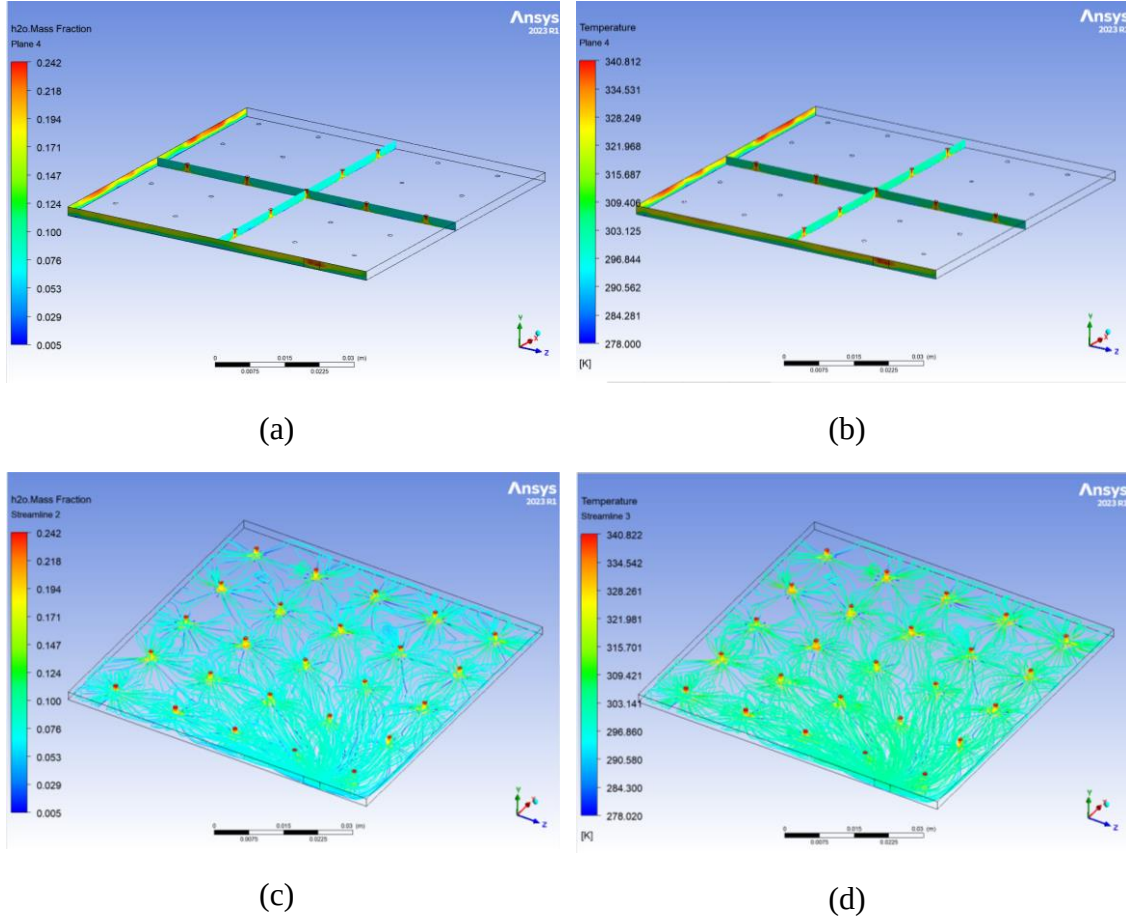


Figure 31 Contours of (a) H₂O Mass Fraction and (b) Temperature Distribution on Plane for Jet Impingement Flow at $v = 90$ m/s. Streamline Visualization of (c) H₂O and (d) Temperature Mass Fraction Distribution in Jet Impingement Configuration at $v = 90$ m/s.

Figures 31(a-d) show the flow at 90 m/s, where the flow is fully turbulent. Figure 31(a) shows a uniform H₂O mass fraction distribution due to intense turbulence, ensuring effective moisture coverage. Figure 31(b) displays the sharpest temperature gradients, indicating maximum heat transfer efficiency. Figures 31(c) and (d) reveal chaotic, turbulent flow with strong mixing, optimizing both moisture distribution and heat transfer. These results, marked by the highest Nusselt number, demonstrate how fully developed turbulence maximizes heat and mass transfer efficiency.

4.2.2. Sherwood Number Analysis

Q. Effect of Temperature on Sherwood Number

Figure 32 shows that at the highest temperature (340.8 K), the Sherwood number is 4.532, indicating highly efficient mass transfer. Higher temperatures provide vapor molecules with more energy, enhancing their movement and condensation. As temperature decreases, the Sherwood number also drops, indicating reduced mass transfer efficiency. For example, at 317 K, the Sherwood number is 0.928, reflecting slower condensation due to lower vapor energy. Small variations in the Sherwood number may be caused by factors like flow or surface conditions, but the overall trend shows that higher temperatures improve mass transfer efficiency. In conclusion, Figure 56 demonstrates that increasing temperature enhances the mass transfer process, benefiting condensation and vapor transport applications.

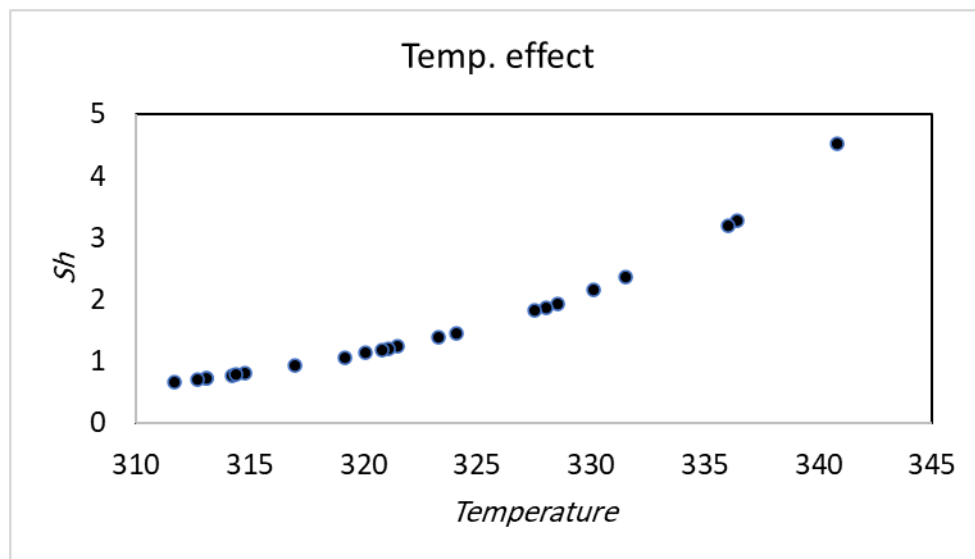


Figure 32 Sherwood Number Vs Temperature.

R. Effect of Condensed Plate Area on Sherwood Number

Figure 33 shows that as surface area decreases, the Sherwood number increases, indicating better mass transfer efficiency for smaller areas due to more concentrated diffusion. However, with larger surface areas, the Sherwood number decreases, not because of reduced mass transfer efficiency, but because the larger area enhances the total mass transfer rate. In summary, smaller areas improve efficiency per unit area, while larger areas increase the total mass transfer rate, highlighting the trade-off between efficiency and total transfer in condensation.

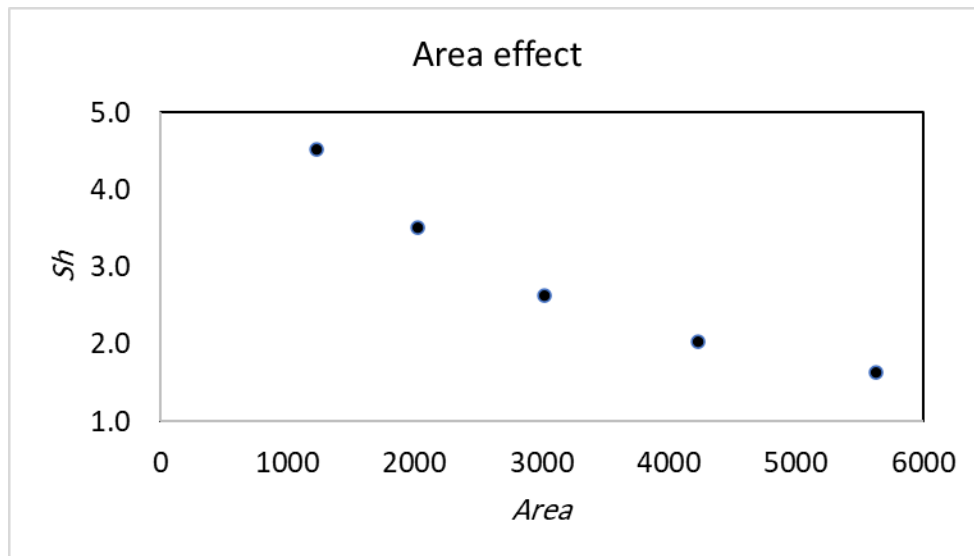


Figure 33 Sherwood Number Vs Area.

S. Effect of S/D on Sherwood Number

Figure 34 depicts the variation of the Sherwood number with changes in the S/D, while keeping all other parameters constant, including vapor temperature, velocity, and H/D ratio. It illustrates the impact of nozzle spacing on mass transfer efficiency. The Sherwood number increases as the S/D ratio rises from 2 to 15, peaking at S/D = 15, which is the best value for mass transfer efficiency. Beyond this point, the Sherwood number either stabilizes or slightly decreases, indicating no further improvement. This trend mirrors jet

array behavior, where optimal spacing maximizes interaction with the surface. Smaller S/D ratios cause excessive jet interference, while larger ratios reduce jet coverage. In summary, an S/D ratio of 15 is best for maximizing the Sherwood number and enhancing mass transfer.

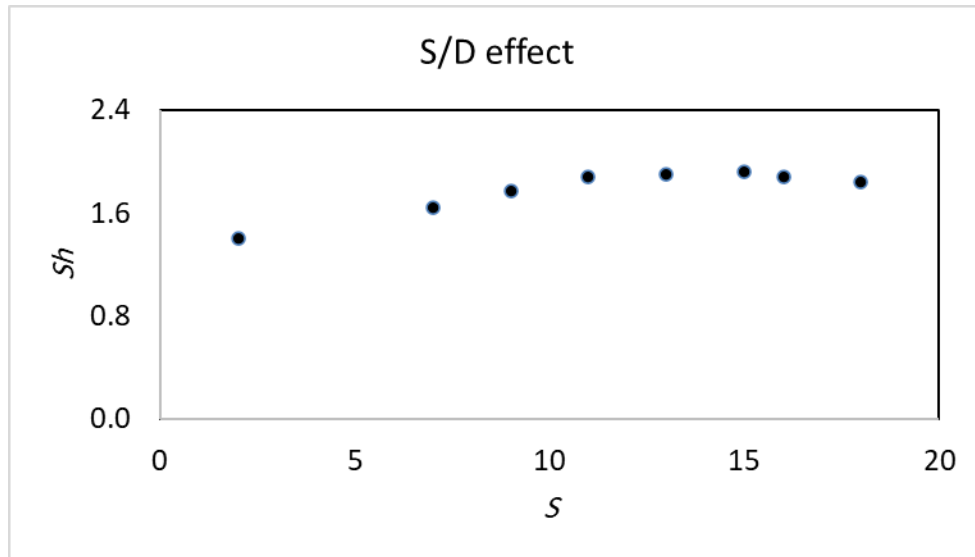


Figure 34 Sherwood Number Vs S/D ratio.

T. Effect of H/D on Sherwood Number

Figure 35 shows that the Sherwood number is highest at an H/D ratio of 2, indicating best mass transfer efficiency due to strong jet-to-surface interaction. As the H/D ratio increases, the Sherwood number gradually decreases, stabilizing below 1.4 at H/D ratios of 10 and above. This trend highlights that smaller H/D ratios enhance jet impingement and mixing, while larger ratios weaken the interaction. In summary, an H/D ratio of 2 is best for maximizing the Sherwood number and effective mass transfer.

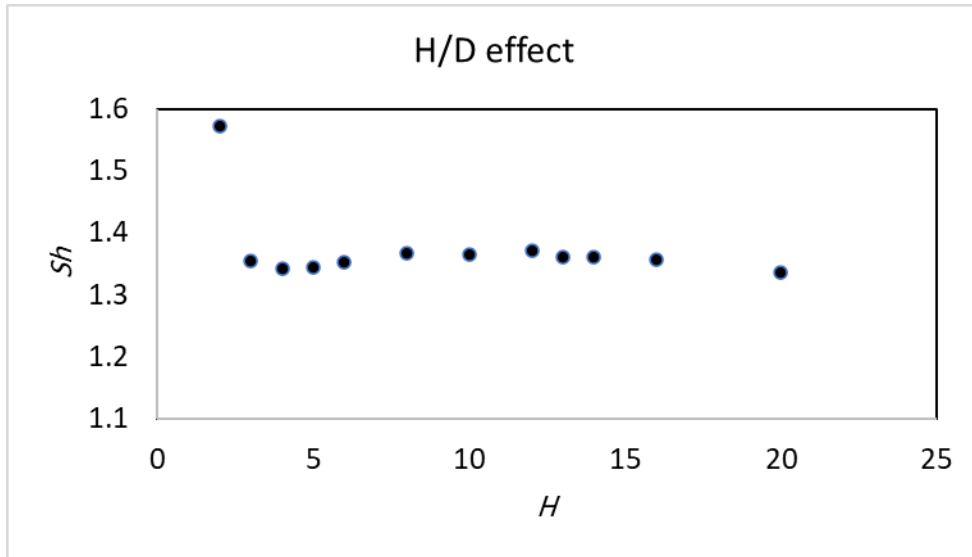


Figure 35 Sherwood Number Vs H/D ratio

U. Effect of Velocity(m/s) on Sherwood Number

Figure 36 shows the relationship between the Sherwood number (Sh) and velocity, illustrating the effect of velocity on mass transfer performance. The x-axis represents the velocity, while the y-axis shows the Sherwood number (Sh).

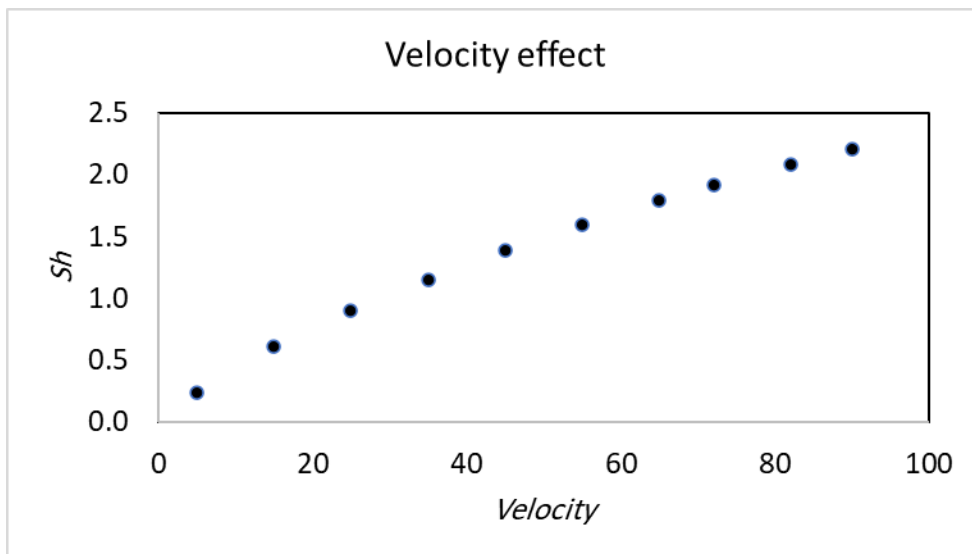


Figure 36 Sherwood Number Vs Velocity (m/s).

As velocity increases from 5 m/s to 90 m/s, the Sherwood number rises from 0.355 to 2.246, due to enhanced turbulence that improves mixing and mass transfer. The non-linear

increase is more pronounced at lower velocities, where laminar flow limits mass transfer efficiency. As velocity increases, the flow becomes turbulent, significantly boosting mass transfer. Overall, the chart shows a direct relationship between velocity and Sherwood number, emphasizing the importance of optimizing flow dynamics for better mass transfer efficiency.

CHAPTER 5

CONCLUSION

This study explored the effectiveness of jet impingement condensation (JIC) systems through experimental and numerical investigations, focusing on optimizing key design parameters to enhance heat and mass transfer for water production. Experimental findings revealed that Configuration 1, featuring a jet-to-plate distance of 2 mm, a plate area of 35 mm x 35 mm, and a nozzle spacing of 7 mm, achieved the highest performance due to concentrated jet impingement. Numerical simulations using ANSYS extended the analysis to a broader parameter space, identifying best values such as an H/D ratio of 2 and an S/D ratio of 15. These configurations significantly improved the Nusselt and Sherwood numbers, highlighting the critical influence of geometric and flow variables on condensation performance.

The results emphasize the importance of minimizing jet-to-plate distance and optimizing nozzle spacing to maximize efficiency. The integration of experimental and numerical approaches provided comprehensive insights into the interplay of parameters affecting JIC systems. This work offers valuable design guidelines for compact, high-performance condensers and demonstrates the potential of JIC technology in industrial applications requiring sustainable water production and efficient thermal management. Future research could explore advanced materials and roughness of condensed plate to further enhance condensation efficiency under varying operational conditions

References

- [1] W. J. Cosgrove and D. P. Loucks, “Water management: Current and future challenges and research directions,” Jun. 01, 2015, *Blackwell Publishing Ltd*. doi: 10.1002/2014WR016869.
- [2] H. Shemer, S. Wald, and R. Semiat, “Challenges and Solutions for Global Water Scarcity,” Jun. 01, 2023, *MDPI*. doi: 10.3390/membranes13060612.
- [3] I. Nabil, A. M. Abdalla, T. M. Mansour, A. I. Shehata, and M. M. K. Dawood, “Salinity impacts on humidification dehumidification (HDH) desalination systems: review,” Jan. 01, 2024. doi: 10.1007/s11356-023-31327-5.
- [4] “Evaporation, Condensation, and the Water Cycle,” 2017. [Online]. Available: <http://www.census.gov/popclock/>.
- [5] V. Novák, “Evapotranspiration: A Component of the Water Cycle,” in *Evapotranspiration in the Soil-Plant-Atmosphere System*, Springer Netherlands, 2012, pp. 1–13. doi: 10.1007/978-94-007-3840-9_1.
- [6] T. Liu and M. S. Mauter, “Heat transfer innovations and their application in thermal desalination processes,” Jun. 15, 2022, *Cell Press*. doi: 10.1016/j.joule.2022.05.004.
- [7] E. S. S. T. APV Americas, “Evaporator Handbook.” [Online]. Available: https://userpages.umbc.edu/~dfrey1/ench445/apv_evap.pdf
- [8] G. Besagni, F. Inzoli, and T. Ziegenhein, “Two-Phase Bubble Columns: A Comprehensive Review”, doi: 10.3390/2020013.
- [9] D. Curto, V. Franzitta, and A. Guercio, “A review of the water desalination technologies,” Jan. 02, 2021, *MDPI AG*. doi: 10.3390/app11020670.
- [10] D. Wickert and G. Prokop, “Simulation of water evaporation under natural conditions—A state-of-the-art overview,” *Exp Comput Multiph Flow*, vol. 3, no. 4, pp. 242–249, Dec. 2021, doi: 10.1007/s42757-020-0071-5.
- [11] “Heat Transfer Process Applications for,” 2004. [Online]. Available: www.cepmagazine.org
- [12] H. T. El-Dessouky, H. M. Ettouney, and Y. Al-Roumi, “Multi-stage flash desalination: present and future outlook,” *Chemical Engineering Journal*, vol. 73, no. 2, pp. 173–190, May 1999, doi: 10.1016/S1385-8947(99)00035-2.

- [13] G. Mittelman, O. Mouchtar, and A. Dayan, “Large-scale solar thermal desalination plants: A review,” Nov. 2007. doi: 10.1080/01457630701421711.
- [14] S. A. El-Agouz and M. Abugderah, “Experimental analysis of humidification process by air passing through seawater,” *Energy Convers Manag*, vol. 49, no. 12, pp. 3698–3703, Dec. 2008, doi: 10.1016/j.enconman.2008.06.033.
- [15] C. Leonard, J. H. Ferrasse, O. Boutin, S. Lefevre, and A. Viand, “Bubble column reactors for high pressures and high temperatures operation,” *Chemical Engineering Research and Design*, vol. 100, pp. 391–421, Aug. 2015, doi: 10.1016/J.CHERD.2015.05.013.
- [16] P. Rollbusch *et al.*, “Bubble columns operated under industrially relevant conditions – Current understanding of design parameters,” *Chem Eng Sci*, vol. 126, pp. 660–678, Apr. 2015, doi: 10.1016/J.CES.2014.11.061.
- [17] J. Huang, J. Zhang, and L. Wang, “Review of vapor condensation heat and mass transfer in the presence of non-condensable gas,” *Appl Therm Eng*, vol. 89, pp. 469–484, Oct. 2015, doi: 10.1016/J.APPLTHERMALENG.2015.06.040.
- [18] I. Tanasawa, “Advances in Condensation Heat Transfer,” 1991. doi: 10.1016/s0065-2717(08)70334-4.
- [19] C. P. Ribeiro and P. L. C. Lage, “Gas-liquid direct-contact evaporation: A review,” Oct. 2005. doi: 10.1002/ceat.200500169.
- [20] Max, “An Applicable Introduction to different types of Condensers Available online:”, [Online]. Available: <https://www.linquip.com/blog/different-types-of-condensers/#:~:text=A>
- [21] MD Iyas, “Types of Condensers: Their Working & Applications.” [Online]. Available: <https://www.theengineerspost.com/types-of-condensers/>
- [22] AIRAH, “WATER-COOLED (TYPE 2) CONDENSERS,” 2017. [Online]. Available: <https://airah.org.au/Common/Uploaded%20files/Resources/SkillsWorkshop/sw099.pdf>
- [23] A. Chiasson, “Waste heat rejection methods in geothermal power generation,” *Geothermal Power Generation: Developments and Innovation*, pp. 423–442, Jan. 2016, doi: 10.1016/B978-0-08-100337-4.00015-2.
- [24] L. Tieghi, G. Delibra, J. van der Spuy, and A. Corsini, “Leading Edge Bumps for Flow Control in Air-Cooled Condensers †,” *International Journal of*

Turbomachinery, Propulsion and Power, vol. 8, no. 1, Mar. 2023, doi: 10.3390/ijtp8010009.

- [25] R. Hashim, S. Hammdi, and A. Eidan, "Evaporative Cooling: A Review of its Types and Modeling," *Basrah journal for engineering science*, vol. 22, no. 1, pp. 36–47, Apr. 2022, doi: 10.33971/bjes.22.1.5.
- [26] V. Katti and S. V. Prabhu, "Experimental study and theoretical analysis of local heat transfer distribution between smooth flat surface and impinging air jet from a circular straight pipe nozzle," *Int J Heat Mass Transf*, vol. 51, no. 17–18, pp. 4480–4495, Aug. 2008, doi: 10.1016/J.IJHEATMASSTRANSFER.2007.12.024.
- [27] V. A. Chiriac and A. Ortega, "A numerical study of the unsteady flow and heat transfer in a transitional confined slot jet impinging on an isothermal surface," *Int J Heat Mass Transf*, vol. 45, no. 6, pp. 1237–1248, Mar. 2002, doi: 10.1016/S0017-9310(01)00224-1.
- [28] M. Angioletti, R. M. Di Tommaso, E. Nino, and G. Ruocco, "Simultaneous visualization of flow field and evaluation of local heat transfer by transitional impinging jets," *Int J Heat Mass Transf*, vol. 46, no. 10, pp. 1703–1713, May 2003, doi: 10.1016/S0017-9310(02)00479-9.
- [29] F. V. Barbosa, S. F. C. F. Teixeira, and J. C. F. Teixeira, "Convection from multiple air jet impingement - A review," Jan. 05, 2023, *Elsevier Ltd.* doi: 10.1016/j.applthermaleng.2022.119307.
- [30] C. Randall Truman, "Jet array impingement with crossflow-correlation of streamwise resolved flow and heat transfer distributions," 1981. [Online]. Available: <https://www.researchgate.net/publication/24385172>
- [31] N. Zuckerman and N. Lior, "Jet Impingement Heat Transfer: Physics, Correlations, and Numerical Modeling," *Adv Heat Transf*, vol. 39, no. C, pp. 565–631, Jan. 2006, doi: 10.1016/S0065-2717(06)39006-5.
- [32] R. Chauhan, T. Singh, N. Kumar, A. Patnaik, and N. S. Thakur, "Experimental investigation and optimization of impinging jet solar thermal collector by Taguchi method," *Appl Therm Eng*, vol. 116, pp. 100–109, Apr. 2017, doi: 10.1016/J.APPLTHERMALENG.2017.01.025.
- [33] H. Martin, "Heat and Mass Transfer between Impinging Gas Jets and Solid Surfaces," *Adv Heat Transf*, vol. 13, no. C, pp. 1–60, Jan. 1977, doi: 10.1016/S0065-2717(08)70221-1.

- [34] R. Viskanta, "Nusselt-Reynolds Prize Paper Heat Transfer to Impinging Isothermal Gas and Flame Jets," 1993.
- [35] I. Nastase and F. Bode, "Impinging jets - A short review on strategies for heat transfer enhancement," in *E3S Web of Conferences*, EDP Sciences, Feb. 2018. doi: 10.1051/e3sconf/20183201013.
- [36] P. S. Penumadu and A. G. Rao, "Numerical investigations of heat transfer and pressure drop characteristics in multiple jet impingement system," *Appl Therm Eng*, vol. 110, pp. 1511–1524, Jan. 2017, doi: 10.1016/J.APPLTHERMALENG.2016.09.057.
- [37] A. H. Alenezi, A. Almutairi, H. M. Alhajeri, A. Addali, and A. A. A. Gamil, "Flow structure and heat transfer of jet impingement on a rib-roughened flat plate," *Energies (Basel)*, vol. 11, no. 6, Jun. 2018, doi: 10.3390/en11061550.
- [38] R. Ben Kalifa, S. Habli, N. M. Saïd, H. Bournot, and G. Le Palec, "Parametric analysis of a round jet impingement on a heated plate," *Int J Heat Fluid Flow*, vol. 57, pp. 11–23, Feb. 2016, doi: 10.1016/j.ijheatfluidflow.2015.11.005.
- [39] S. Yong, J. Z. Zhang, and G. N. Xie, "Convective heat transfer for multiple rows of impinging air jets with small jet-to-jet spacing in a semi-confined channel," *Int J Heat Mass Transf*, vol. 86, pp. 832–842, Jul. 2015, doi: 10.1016/j.ijheatmasstransfer.2015.03.073.
- [40] B. Weigand and S. Spring, "Multiple Jet Impingement-A Review," 2011.
- [41] P. Hrycak and S. Snaw, "HEAT TRANSFER FROM IMPINGING JETS A LITERATURE REVIEW Sill 02279," 1981.
- [42] D. G. ARGANBRIGHT and H. RESCH, "A Review of Basic Aspects of Heat Transier under Impinging Air Jets," Springer-Verlag, 1971.
- [43] B. W. Webb and C.-F. Ma, "Single-Phase Liquid Jet Impingement Heat Transfer," 1995.
- [44] A. Sarkar, N. Nitin, M. V Karwe, and R. P. Singh, "Fluid Flow and Heat Transfer in Air Jet Impingement in Food Processing," 2004. [Online]. Available: www.ift.org
- [45] S. Pawar and D. K. Patel, "The Impingement Heat Transfer Data of Inclined Jet in Cooling Applications: A Review," Feb. 01, 2020, *Science Press*. doi: 10.1007/s11630-019-1200-y.

- [46] V. Katti and S. V. Prabhu, "Influence of spanwise pitch on local heat transfer distribution for in-line arrays of circular jets with spent air flow in two opposite directions," *Exp Therm Fluid Sci*, vol. 33, no. 1, pp. 84–95, Oct. 2008, doi: 10.1016/J.EXPTHERMFLUSCI.2008.07.004.
- [47] L. Gao, S. V. Ekkad, and R. S. Bunker, "Impingement heat transfer, part I: Linearly stretched arrays of holes," *J Thermophys Heat Trans*, vol. 19, no. 1, pp. 57–65, 2005, doi: 10.2514/1.8551.
- [48] W. Li, M. Xu, J. Ren, and H. Jiang, "Experimental Investigation of Local and Average Heat Transfer Coefficients Under an Inline Impinging Jet Array, Including Jets With Low Impingement Distance and Inclined Angle," *J Heat Transfer*, vol. 139, no. 1, Jan. 2017, doi: 10.1115/1.4034165.
- [49] C. R. Truman, D. E. Metzger Professor Mem ASME, and M. -----Willi, "Streamwise Flow and Heat Transfer Distributions for Jet Array Impingement with Crossflow Graduate Student Student Mem. ASME," 1981. [Online]. Available: <http://asmedigitalcollection.asme.org/GT/proceedings-pdf/GT1981/79634/V003T09A005/2393497/v003t09a005-81-gt-77.pdf>
- [50] M. Goodro, J. Park, P. Ligrani, M. Fox, and H. K. Moon, "Effects of hole spacing on spatially-resolved jet array impingement heat transfer," *Int J Heat Mass Transf*, vol. 51, no. 25–26, pp. 6243–6253, Dec. 2008, doi: 10.1016/j.ijheatmasstransfer.2008.05.004.
- [51] Y. Xing, S. Spring, and B. Weigand, "Experimental and numerical investigation of heat transfer characteristics of inline and staggered arrays of impinging jets," *J Heat Transfer*, vol. 132, no. 9, pp. 1–11, Sep. 2010, doi: 10.1115/1.4001633.
- [52] N. Celik and E. Turgut, "Design analysis of an experimental jet impingement study by using Taguchi method," *Heat and Mass Transfer/Waerme- und Stoffuebertragung*, vol. 48, no. 8, pp. 1407–1413, Aug. 2012, doi: 10.1007/s00231-012-0989-7.
- [53] S. Caliskan, S. Baskaya, and T. Calisir, "Experimental and numerical investigation of geometry effects on multiple impinging air jets," *Int J Heat Mass Transf*, vol. 75, pp. 685–703, 2014, doi: 10.1016/j.ijheatmasstransfer.2014.04.005.
- [54] Y. Ichikawa, M. Motosuke, Y. Kameya, M. Yamamoto, and S. Honami, "Three-dimensional flow characterization of a square array of multiple circular impinging jets using stereoscopic PIV and heat transfer relation," *J Vis (Tokyo)*, vol. 19, no. 1, pp. 89–101, Feb. 2016, doi: 10.1007/s12650-015-0296-8.

- [55] R. Chauhan, T. Singh, N. Kumar, A. Patnaik, and N. S. Thakur, "Experimental investigation and optimization of impinging jet solar thermal collector by Taguchi method," *Appl Therm Eng*, vol. 116, pp. 100–109, 2017, doi: 10.1016/j.applthermaleng.2017.01.025.
- [56] V. Katti and S. V. Prabhu, "Influence of spanwise pitch on local heat transfer distribution for in-line arrays of circular jets with spent air flow in two opposite directions," *Exp Therm Fluid Sci*, vol. 33, no. 1, pp. 84–95, Oct. 2008, doi: 10.1016/j.expthermflusci.2008.07.004.
- [57] A. M. Huber and R. Viskantay, "Effect of jet-jet spacing on convective heat transfer to confined, impinging arrays of axisymmetric air jets," 1994.
- [58] J. M. Buchlin, "Convective heat transfer in impinging- gas- jet arrangements," *Journal of Applied Fluid Mechanics*, vol. 4, no. 3, pp. 137–149, 2011, doi: 10.36884/jafm.4.02.11926.
- [59] M. Draksler, B. Končar, L. Cizelj, and B. Ničeno, "Large Eddy Simulation of multiple impinging jets in hexagonal configuration – Flow dynamics and heat transfer characteristics," *Int J Heat Mass Transf*, vol. 109, pp. 16–27, 2017, doi: 10.1016/j.ijheatmasstransfer.2017.01.080.
- [60] F. V. Barbosa, S. D. T. Sousa, S. F. C. F. Teixeira, and J. C. F. Teixeira, "Application of Taguchi Method for the Analysis of a Multiple Air Jet Impingement System with and without Target Plate Motion," *Int J Heat Mass Transf*, vol. 176, Sep. 2021, doi: 10.1016/j.ijheatmasstransfer.2021.121504.
- [61] Z. W. P. I. and T. V. Jones. K. W. Van Treuren, "*Comparison and Prediction of Local and Average Heat Transfer Coefficients Under an Array of Inline and Staggered Impinging Jets*," in *International Gas Turbine and Aemengine Congress & Exhibition*. 1996.
- [62] S. Pachpute and B. Premachandran, "Turbulent multi-jet impingement cooling of a heated circular cylinder," *International Journal of Thermal Sciences*, vol. 148, Feb. 2020, doi: 10.1016/j.ijthermalsci.2019.106167.
- [63] S. Caliskan and S. Baskaya, "Experimental investigation of impinging jet array heat transfer from a surface with V-shaped and convergent-divergent ribs," *International Journal of Thermal Sciences*, vol. 59, pp. 234–246, Sep. 2012, doi: 10.1016/j.ijthermalsci.2012.04.013.
- [64] G. E. Andrews, R. A. A. A. Hussain, and M. C. Mkpadi, "Enhanced Impingement Heat Transfer: Comparison of Co-flow and Cross-flow with Rib Turbulators," 2003.

- [65] Y. Xing and B. Weigand, “Experimental investigation of impingement heat transfer on a flat and dimpled plate with different crossflow schemes,” *Int J Heat Mass Transf*, vol. 53, no. 19–20, pp. 3874–3886, Sep. 2010, doi: 10.1016/j.ijheatmasstransfer.2010.05.006.
- [66] P. M. Ligrani, Z. Ren, and W. C. Buzzard, “Impingement jet array heat transfer with small-scale cylinder target surface roughness arrays,” *Int J Heat Mass Transf*, vol. 107, pp. 895–905, 2017, doi: 10.1016/j.ijheatmasstransfer.2016.10.123.
- [67] A. Chitsazan, G. Klepp, B. Glasmacher, and K. M. Pour, “Numerical optimization of drying energy consumption from multiple jets impinging on a moving curved surface,” *International Journal of Heat and Technology*, vol. 39, no. 3, pp. 755–762, Jun. 2021, doi: 10.18280/ijht.390309.
- [68] L. Hussain *et al.*, “Heat transfer augmentation through different jet impingement techniques: A state-of-the-art review,” *Energies (Basel)*, vol. 14, no. 20, Oct. 2021, doi: 10.3390/en14206458.
- [69] Y. Cheng and Z. Wang, “New approach for efficient condensation heat transfer,” Mar. 01, 2019, *Oxford University Press*. doi: 10.1093/nsr/nwy161.
- [70] S. Bortolin, M. Tancon, and D. Del Col, “Heat Transfer Enhancement During Dropwise Condensation Over Wettability-Controlled Surfaces,” in *The Surface Wettability Effect on Phase Change*, M. Marengo and J. De Coninck, Eds., Cham: Springer International Publishing, 2022, pp. 29–67. doi: 10.1007/978-3-030-82992-6_3.
- [71] J. Dong, S. Lu, B. Liu, J. Wu, and M. Chen, “Numerical Investigation of Heat Transfer and Development in Spherical Condensation Droplets,” *Micromachines (Basel)*, vol. 15, no. 5, May 2024, doi: 10.3390/mi15050566.
- [72] L. Bogdanic, H. Auracher, and F. Ziegler, “Two-phase structure above hot surfaces in jet impingement boiling,” in *Heat and Mass Transfer/Waerme- und Stoffuebertragung*, May 2009, pp. 1019–1028. doi: 10.1007/s00231-007-0272-5.
- [73] L. de L. G. M. C. C. Meola, “Azimuthal instability in an impinging jet: adiabatic wall temperature distribution,” 1995.
- [74] J. Badra, A. R. Masri, and M. Behnia, “Enhanced transient heat transfer from arrays of jets impinging on a moving plate,” *Heat Transfer Engineering*, vol. 34, no. 4, pp. 361–371, Jan. 2013, doi: 10.1080/01457632.2013.717046.
- [75] R. Vinze, S. Chandel, M. D. Limaye, and S. V. Prabhu, “Influence of jet temperature and nozzle shape on the heat transfer distribution between a smooth

- plate and impinging air jets,” *International Journal of Thermal Sciences*, vol. 99, pp. 136–151, Jan. 2016, doi: 10.1016/j.ijthermalsci.2015.08.009.
- [76] P. Grenson, O. Léon, P. Reulet, and B. Aupoix, “Investigation of an impinging heated jet for a small nozzle-to-plate distance and high Reynolds number: An extensive experimental approach,” *Int J Heat Mass Transf*, vol. 102, pp. 801–815, Nov. 2016, doi: 10.1016/j.ijheatmasstransfer.2016.06.076.
- [77] M. Fénot, X. T. Trinh, and E. Dorignac, “Flow and heat transfer of a compressible impinging jet,” *International Journal of Thermal Sciences*, vol. 136, pp. 357–369, Feb. 2019, doi: 10.1016/j.ijthermalsci.2018.10.035.
- [78] P. A. K. Lam and K. A. Prakash, “A numerical investigation and design optimization of impingement cooling system with an array of air jets,” *Int J Heat Mass Transf*, vol. 108, pp. 880–900, 2017, doi: 10.1016/j.ijheatmasstransfer.2016.12.017.
- [79] J.-Y. San and M.-D. Lai, “Optimum jet-to-jet spacing of heat transfer for staggered arrays of impinging air jets,” 2001. [Online]. Available: www.elsevier.com/locate/ijhmt
- [80] P. S. Penumadu and A. G. Rao, “Numerical investigations of heat transfer and pressure drop characteristics in multiple jet impingement system,” *Appl Therm Eng*, vol. 110, pp. 1511–1524, Jan. 2017, doi: 10.1016/j.applthermaleng.2016.09.057.
- [81] P. Rollbusch *et al.*, “Bubble columns operated under industrially relevant conditions - Current understanding of design parameters,” Apr. 04, 2015, *Elsevier Ltd*. doi: 10.1016/j.ces.2014.11.061.
- [82] S. A. El-Agouz and M. Abugderah, “Experimental analysis of humidification process by air passing through seawater,” *Energy Convers Manag*, vol. 49, no. 12, pp. 3698–3703, Dec. 2008, doi: 10.1016/j.enconman.2008.06.033.
- [83] A. S. L. P. I. and D. P. D. THEODORE L. BERGMAN, “Fundamentals of Heat and Mass Transfer,” pp. 477–482, 2011.

Vitae

Name : Turki Shaya Al Ati

Nationality : Saudi

Date of Birth : 9/7/1990

Email : turki_sh@icloud.com

Address : Alnawras Street, Building 223
Jubail, Eastern Region
Saudi Arabia
Postal Code: 35412

Academic Background : Master of Science (MSc), Dec 2024

Mechanical Engineering Department

King Fahad University of Petroleum & Minerals

Dhahran, Saudi Arabia

Bachelor of Mechanical Engineering Technology, Oct 2015

Jubail Industrial College

Jubail, Saudi Arabia

Research Interests : Jet Impingement Condensation, Heat and Mass Transfer
and Desalination.