People's Democratic Republic of Algeria
Ministry of Higher Education and Scientific Research
Higher Normal School of Technological Education
- Skikda -

Technology Department
Speciality: Mechanical Engineering

Graduation Thesis for Obtaining a High School Teaching Certificate.

Title

Improving Heat Transfer in Vertical and Horizontal Tubes

Prepared by :

- Bettiche Bouthaina
- Atamnia Abir

Under the supervision of the committee :

- President: Dr. Bouhali Rima — Professor lecturer b
- Supervisor: Dr. Merimech Imane — Professor lecturer
- Examiner: Dr. Ben Sanaa Toufik — Professor lecturer b

Academic year: 2023\2024

GRATITUDE

We want to thank God first and foremost, the Almighty, for giving us the courage, patience, and determination to achieve this goal. I would like to express my gratitude and respect today to my dear and esteemed teachers, especially Dr.Imene Merimeche, who has always been understandable and available throughout the preparation of this research, for the support, assistance, and time she has given us, and for the trust, encouragement, and valuable advice she has provided.

We also thank the members of the examination committee. We would also like to thank Dr. Rima Bouhali, the chairperson of the examination committee for this research, and the examiner Dr. Toufik Ben Sanaa, who honored us by accepting to be the examiners for this work.

We express our deep thanks and gratitude to all the professors of the Mechanical Engineering Department.

We want to thank all the staff of the Higher School of Teachers in Skikda, especially the head of the Technology Department, Professor Salih Rahmoni, and the school director Djamal Boudjaadare.

We will not forget all those who contributed in any way to the completion of this work, whether through their skills, technical assistance, or advice.

Special thanks to the English teacher Ouchnane Mounia.

DEDICATION

Oh Allah, the night does not become peaceful except through thanking You, and the day does not become pleasant except through obeying You, and the moments do not become enjoyable except through remembering You, and Paradise does not become delightful except through seeing You.

To the one who delivered the message, fulfilled the trust, and advised the nation, to the Prophet of mercy and light of the worlds, our master Muhammad, peace upon him.

To the one whom Allah adorned with dignity and honor, and to the one who carries his name with pride, my dear father "Bettiche Omar".

To my angel in life, to the meaning of love and tenderness, to the one whose prayers are the secret of my success, my beloved mother "Abadi Djamila". I ask Allah to prolong their lives to see the fruits of their hardworking.

To those who shared the burdens of life with me, and distinguished themselves with generosity, to those in whose eyes I see optimism and in whose smiles I see happiness, my siplings "Salaheddine, Mouhieddine, Yaakoub, Rahma, and Salsabil".

To the joy of my soul and the delight of my eyes, to the love of my heart and the source of my happiness, my dear grandfather "Bettiche Belgacem".

To my sister, my friend, my beloved, and my heart, "Bouzidi Marwa" , may Allah have mercy on you and admit you to His paradise, and make our eternal meeting in the paradise of eternity.

To my companion on the path and my sister, not from the same mother, but the same soul, to my great source of strength and love, to those who shared my research, effort, and many nights of vigil," Atamnia Abir" .

To those who excelled in generosity and distinguished themselves with loyalty, to those with whom I found joy and happiness in the sweet and bitter paths of life"Amina, Fatima, Zouina, Nessrin, Aya, Fatma".

To my companions in my exileto those with whom I spent the most beautiful and enjoyable moments and years," Aya, Hadil, Rayan, Marwa, Aya Bouslama, Aya saker , Ahlam, Samah". To those who were a support and a light on my path.

To my teachers in all educational life, to my colleagues at the Higher School of Technological Education in Skikda. To all of these, I dedicate this work.

Bettiche Bouthaina

DEDICATION

I dedicate my work:

To my support, my strength, my hope in this world, my father"Khemissi"

To my joy, my happiness, my supporter, my friend, my mother"Atamnia Akila"

To my heart, my beloved ones, the reason for my joy and happiness in this life, my siblings"Amine" "Chahinez"and "Abd ElAli".

To the spirit of my lost one, whose memory lives in my heart, my grandfathers"A.Mohammed".

To my grandmothers, U ncles, Aunts, Maternal aunts .and all my family.

To those whose support comes to me in my hardest moments and in my happiest times.my cousins "Djalel", "Anise","Amira", "Ahlem","Chaima"and "Dhikra".

To my companion on the path and my sister who was not born to my mother, to my great repository of strength and love, to those who shared my research effort, and many nights of vigil"Bettiche Bouthaina"

To my friends, the balm of my heart and the joy of my soul, despite the distance between us, they are the delight of my days" Radia","Imane","Hana","Chaima","Youssra","Youssra".

To those who eased the bitterness of days by their presence, with whom I shared sweet and bitter moments, to those who brightened my days, my companions "Douaa, Safa, Ahlem, Maroua, Nada, Lamia, Hadil, Aya, Aya, Youssra, Malak, Ikrame".

To everyone who taught me a letter throughout my journey. To my fellow classmates.

ATamnia Abir

Abstract

This research addresses the highly important topic of "Enhancing Heat Transfer in Horizontal and Vertical Pipes," within the field of thermal engineering. The aim of this study is to investigate the factors influencing heat transfer efficiency and to develop techniques for its improvement under the influence of forced internal heat load.

Using simulation and numerical modeling software Gambit and Fluent, the study provided an opportunity to analyze the thermal behavior of pipes with high precision. The Finite Volume Method was adopted for heat transfer efficiency analysis by solving equations based on the k−ε model.

Experimental results revealed that in both horizontal and vertical pipes, with and without control elements at various positions, heat transfer efficiency in the laminar system is superior when employing control elements at three positions. Conversely, in the turbulent system, natural heat transfer efficiency is better than when control elements are added. It was also concluded that heat transfer efficiency in the turbulent system is superior to the laminar system. Finally, the use of control elements in horizontal and vertical pipes enhances heat transfer.

Keywords: Heat transfer, forced internal heat load, horizontal and vertical pipes, Finite Volume Method, Fluent and Gambit, control elements.

Résumé

Cette recherche aborde un sujet crucial en ingénierie thermique "Amélioration du transfert de chaleur dans les tubes horizontaux et verticaux". L'objectif de cette étude est d'examiner les facteurs influençant l'efficacité du transfert de chaleur et de développer des techniques pour l'améliorer sous l'effet de la charge thermique interne forcée.

En utilisant les logiciels de simulation et de modélisation numérique Gambit et Fluent, l'étude a offert une opportunité d'analyser précisément le comportement thermique des tubes. La méthode des volumes finis a été adoptée pour analyser l'efficacité du transfert de chaleur en résolvant les équations basées sur le modèle k-ε.

Les résultats expérimentaux ont montré que dans les tubes horizontaux et verticaux, avec et sans éléments de contrôle à différentes positions, l'efficacité du transfert de chaleur est meilleure dans le système laminaire lorsqu'on utilise des éléments de contrôle à trois positions. En revanche, dans le système turbulent, l'efficacité du transfert de chaleur naturel est meilleure que lorsque des éléments de contrôle sont ajoutés. De plus, il a été conclu que l'efficacité du transfert de chaleur est supérieure dans le système turbulent par rapport au système laminaire. Enfin, l'utilisation d'éléments de contrôle dans les tubes horizontaux et verticaux améliore le transfert de chaleur.

Mots-clés : Transfert de chaleur, charge thermique interne forcée, tubes horizontaux et verticaux, méthode des volumes finis, Fluent et Gambit, éléments de contrôle.

Abstract

ملخص

يتناول هذا البحث "تحسين انتقال الحرارة في الانابيب الافقية والعمودية " موضوعا بالغ الاهمية في مجال هندسة الحرارة، يهدف هذا البحث الى دراسة العوامل المؤثرة على كفاءة نقل الحرارة وتطوير تقنيات تحسينها تحت تأثير الحمل الحراري القسري الداخلي.

باستخدام برامج المحاكاة والنمذجة العددية Gambit وFluent، اتاحت الدراسة فرصة لتحليل السلوك الحراري للأنابيب بدقة عالية، وتم اعتماد طريقة الحجوم المنتهية في الحسابات لتحليل كفاءة انتقال الحرارة لحل المعادلات التي ترتكز على نموذج k-ε .

ومن خلال النتائج التجريبية تبين انه في الانابيب الافقية والعمودية بدون و مع عناصر تحكم في مواضع مختلفة حيث انه في النظام الصفائحي تكون كفاءة انتقال الحرارة في الانابيب احسن عند توظيف عناصر التحكم في ثلاث مواضع ، اما بالنسبة للنظام المضطرب تكون كفاءة انتقال الحرارة في الحالة الطبيعية احسن من إضافة عناصر التحكم ، كما توصلنا الى ان كفاءة انتقال الحرارة في النظام المضطرب احسن من الصفائحي ، و أخيرا فان توظيف عناصر التحكم في الانابيب الافقية و العمودية يحسن من انتقال الحرارة .

الكلمات المفتاحية: انتقال الحرارة، الحمل الحراري القسري، الانابيب الافقية و العمودية ، الحجوم المنتهية، Fluent و Gambit، عناصر التحكم.

Table of Contents

Table des matières

Figure list 1

Figure list 4

Nomenclature 5

General Introduction : 1

Chapter I : 2

General Information on Heat Transfer and fluid mecanique in Tubes 2

I. 1 Introduction 3

I. 2 Heat transfer mechanisms : 3

I. 2. 1 Conduction : 3

I. 2. 2 Convection : 3

I. 2. 3 Radiation : 5

I. 3 Dimensionless Number in Natural Convection : 5

I. 4 Dimensionless Number in Forced Convection 6

I. 5 Classification of fluid flow. 8

I. 5. 1 Steady, Unsteady Flow 8

I. 5. 2 Laminar flow and Turbulent flow 8

I. 5. 3 Compressible versous Incompressible Flow 8

I. 5. 4 Viscous versus Inviscid Flow 8

I. 5. 5 Natural flow versus forced flow 8

I. 5. 6 Internal versus External flow 9

I. 5. 7 One-Two and Three dimension flow 9

I. 6 Velocity Boundry Layer 10

I. 7 Thermal Boundary Layer 11

I. 8 Laminar flows and Turbulent flows 12

I. 9 Internal Forced Convection 12

I. 9. 1 Average Velocity and Temperature 13

I. 9. 2 The Entrance Region 14

I. 9. 3 The Fully Devoloped Region 14

I. 9. 4 General Thermal Analysis 14

I. 9. 5 Laminer flow in tube 14

I. 9. 6 Turbulent flow in tube 16

I. 10 Heat transfer enhancement technique 16

I. 10. 1 Passive method 17

Table of Contents

I. 10. 2 Active method 17

I. 11 Vortex Generator 17

I. 12 Conclusion 18

Chapter II :Molding and Numerical Resolution Procedure 19

II. 1 Introduction 20

II. 2 Conservation equation 20

II. 2. 1 Equation of quantity of motion 20

II. 2. 2 Continuity equation 21

II. 2. 3 Energy equation 21

II. 3 Presentation of the GAMBIT Software 21

II. 3. 1 Starting GAMBIT 22

II. 3. 2 GAMBIT Features 22

II. 3. 3 Description of Small Geometry in Gambit 23

II. 3. 4 Description of the mesh menu 24

II. 3. 5 Boundary conditions and definition of domains 25

ll. 3. 6 Display Menu 26

II. 3. 7 Export of the gambit mesh 27

ll. 4 Fluent software 28

II. 4. 1 Verification of the mesh matters 29

II. 4. 2 Definition of fluid characteristics 30

II. 4. 3 Introduction of gravity 31

II. 4. 4 Condition at limit 32

II. 5 The finite volume method (FVM) 35

II. 5. 1 Introduction 35

II. 5. 2 Definition of (FVM) 35

II. 6 The Turbulence Phenomenon 38

II. 6. 1 Introduction 38

II. 6. 2 Definition of Turbulence Phenomenon 38

II. 6. 3 General characteristics of turbulence 38

II. 6. 4 K-ε turbulence model : 39

II. 7 The Trapezius Method 40

II. 7. 1 Introduction 40

II. 7. 2 Definition 40

II. 7. 3 Objective of the Method: 40

II. 8 Conclusion 40

Table of Contents

Chapter III : 42
Results and Interpretations 42
III. 1 Introduction 43
III. 2 Problem Statement 43
III. 3 Mesh Selection 44
III. 4 Flow control for thermal transfer enhancement 47
III. 4. 1 Smooth vertical and horizontal conduits 47
III. 4. 2 Study of control elements 50
III. 5 Conclusion: 58
General Conclusion : 60
BIBLIOGRAPHIC REFERENCES 61

Figure List

Figure list

Figure I.1	Natural convection.	**09**
Figure I.2	Forced convection.	**10**
Figure I.3	Internal versus External flow.	**14**
Figure I.4	Velocity Boundary Layer In Flat Plate.	**15**
Figure I.5	The development of the velocity boundary layer in a tube. (The developed mean velocity profile will be parabolic in laminar flow, as shown, but somewhat blunt in turbulent flow.	**15**
Figure I.6	Thermal Boundry Layer In Flat plate.	**16**
Figure I.7	Thermal Boundary Layer In Tube.	**16**
Figure I.8	Laminar and turbulent flow regimes of candle.	**17**
Figure I.9	Internal forced convection.	**18**
Figure I.10	Nusselt number and friction factor for fully developed laminar flow in tubes ofvarious cross sections ($\mathbf{D_h}=4\mathbf{A_c}/\boldsymbol{\rho}$, Re=$\mathbf{V_m D_h}$ /v, and Nu= $\mathbf{hD_h}/\lambda$).	**20**
Figure II.1	The gambit guide.	**27**
Figure II.2	Launch of the gambit.	**27**
Figure II.3	Main functions of the Gambit general menu.	**28**
Figure II.4	Secondary Menus of Gambit.	**28**
Figure II.5	Table Description of commands in the geometry menu.	**29**
Figure II.6	Table Description of commands in the mesh menu.	**29**
Figure II.7	Main mesh menus in Gambit.	**30**
Figure II.8	Main menus for boundary conditions.	**31**
Figure II.9	Gambit Display Menus.	**32**
Figure II.10	Exporting the mesh from Gambit.	**33**
Figure II.11	To start the simulation you need to import the file.	**34**
Figure II.12	Verification of Fluent sou mesh.	**34**
Figure II.13	Definition of fluid characteristics.	**35**

Figure List

Figure II.14	Fluid List Interface.	**35**
Figure II.15	Solid List Interface.	**36**
Figure II.16	Introduction of gravity.	**36**
Figure II.17	Limit Condition Values.	**37**
Figure II.18	Input Speed (velocity inlet).	**37**
Figure II.19	Pressure inlet.	**38**
Figure II.20	Mass Flow Inlet.	**38**
Figure II.21	Pressure Outlet.	**39**
Figure II.22	Out flow.	**39**
Figure II.23	Output pressure interface.	**40**
Figure II.24	Discretion in finite volumes of the area of study in the two-dimensional case.	**41**
Figure III.1	conduit with control element (x=50,200,400) and x=200 mm.	**48**
Figure III.2	Chosen Mesh.	**50**
Figure III.3	Subdivision of the conduit section for the calculation of the local Nusselt number.	**50**
Figure III.4	Local Nusselt numbers in a horizontal tube calculated by Fluent compared to the average Nusselt number determined by the correlation of Sahah and London.	**51**
Figure III.5	Local convective heat transfer coefficient in a smooth vertical tube for a turbulent regime.	**52**
Figure III.6	Local convective heat transfer coefficient in a smooth vertical tube for a turbulent regime.	**53**
Figure III.7	Local convective heat transfer coefficient in a smooth horizontal tube for a turbulent regime.	**53**
Figure III.8	Temperature fields for a smooth vertical conduit for laminar regime (a) and turbulent regime (b).	**54**
Figure III.9	Velocity fields for a smooth vertical conduit for laminar regime (a) and turbulent regime (b).	**55**
Figure III.10	Local convective heat transfer coefficient in a vertical and horizontal tube with a control element at x = 200 mm for a laminar regime.	**55**

Figure List

Figure III.11	Local convective heat transfer coefficient in a vertical and horizontal tube with a control element at x = 200 mm for a turbulent regime.	**56**
Figure III.12	Temperature and velocity fields for a horizontal conduit with a control element at x = 200 mm for a laminar regime.	**57**
Figure III.13	Comparison of the variation of the local convective coefficient in a smooth horizontal tube and with a control element at x = 200 mm for a laminar regime.	**57**
Figure III.14	Local convective heat transfer coefficient in a vertical and horizontal tube with control elements at (x = 50, 200, and 400 mm) for a laminar regime.	**58**
Figure III.15	Temperature field for laminar flow in a vertical tube with control elements at (x = 50, 200, and 400 mm).	**59**
Figure III.16	Local convective heat transfer coefficient in a vertical tube with control elements at (x = 50, 200, and 400 mm) for turbulent flow.	**59**
Figure III.17	: Local convective heat transfer coefficient in a horizontal tube with control elements at (x = 50, 200, and 400 mm) for turbulent flow.	**60**
Figure III.18	Variation of the local heat transfer coefficient in three horizontal tubes for laminar flow.	**61**
Figure III.19	Variation of the local heat transfer coefficient in three vertical tubes for laminar flow.	**62**
Figure III.20	Variation of the local heat transfer coefficient in three vertical tubes for turbulent flow.	**63**

Table List

Table list

Table I .1	Typical ranges of Prandtl numbers for common fluids.	**12**
Table II. 1	K-ε turbulence model coefficients.	**45**
Table III.1	Details of the different meshes tested.	**49**
Table III.2	Average Convective Heat Transfer Coefficient for the Different Meshe.	**49**

Nomenclature

Nomenclature

Φ	heat flux	**W**
h	Convenction heat transfer coefficient	$W/m^2.K$
S	Heat transfer surface area	m^2
T	Tempurature	**K**
Gr	Grashof number	
g	the acceleration due to gravity	m/s^2
β	The coefficient of thermal expansion	
L	The characteristic length	**M**
υ	The kinematic viscosity	Pa/s
Ra	Rayleigh number	
Pr	Prandtl number	
Nu	Nusselt number	
Re	Reynolds number	
ρ	The fluid density	kg/m^3
D	The daiametre	**M**
μ	The dynamic viscosity	$kg/m.s$
C_p	The specific heat capacity	
λ	The thermal coductivity	$W/m.K$
V	The velosity	m/s
A_c	The cross section	m^2
U_0	The velocity	m/s
$\dot{m}$	The mass flux	Kg/s
δ	Kronecker delta	
$\dot{E}$	The energy transfered	**W**
$\dot{Q}$	The energy conservation	**W**
$\dot{q}_s$	The surface heat flow	W/m^2
f	Friction coefficient	
R	Radius	**M**
X	Distance	**M**
V_m	The mean velocity	m/s
P	The pressure	**Bar**
K	Turbulent kintic energy	
ε	The dissipation rate	
G_k	Term for the production of turbulent kinetic energy	Kg/ms^3
$\vec{F}$	Force vector	

General Introduction :

The issue of improving heat transfer in horizontal and vertical tubes is one of the fundamental topics in thermal engineering, directly impacting the efficiency of thermal systems in various industrial applications. This study aims to address this vital subject by exploring the factors affecting heat transfer efficiency and developing effective techniques to enhance it under the influence of internal forced convection.

This research highlights the significant importance of improving the performance of thermal systems by providing advanced technical solutions that enhance heat transfer efficiency, thereby boosting the overall performance of thermal systems in various industrial applications. By focusing on horizontal and vertical tubes, a deeper understanding of thermal processes can be achieved, leading to the development of new methods for improvement.

Simulation and Simulation and numerical modeling programs such as Gambit and Fluent were used to accurately analyze the thermal behavior of the tubes. The study relies on the finite volume method in calculations to analyze heat transfer efficiency, allowing for a precise understanding of complex thermal phenomena within the tubes. Through these tools, a variety of scenarios and different cases can be simulated and analyzed to achieve tangible improvements in thermal performance.

Experimental results have shown that the use of control elements significantly improves the effectiveness of heat transfer in horizontal and vertical tubes. These improvements include increased efficiency of thermal systems, contributing to energy consumption reduction and overall performance enhancement. The results provide practical evidence of the proposed methods' effectiveness, allowing for their wide application in the industry.

The results of this research can be applied in a wide range of industrial fields that rely on thermal systems, such as petrochemical industries, power generation, and food and beverage manufacturing. By improving heat transfer, substantial energy savings and operational efficiency improvements can be achieved, enhancing competitiveness and innovation in these industries.

This research constitutes an important contribution to the field of thermal engineering by offering new techniques and methods to improve heat transfer in horizontal and vertical tubes. Relying on numerical modeling and precise analysis, significant improvements in thermal system efficiency can be achieved, contributing to enhanced performance and efficiency in various industrial applications. We hope this study serves as a starting point for further research and development in this vital field.

Chapter I :
General Information on Heat Transfer and fluid mecanique in Tubes

I. 1 Introduction

Enhancing heat transfer is considered as a fundamental element in the continuous development of engineering and technology. Therefore, improving the efficiency of thermal systems promotes sustainability and reduces reliance on other energy sources, leading to increased efficiency and improved overall performance of engineering systems [3].

In this chapter, we will start by introducing the three fundamental mechanisms of heat transfer, including conduction, convection, and radiation. We will then move on to advanced aspects of evaluating and understanding various aspects of natural and forced heat transfer. We will also mention the classification of fluid flows based on their common properties, and then discuss flow velocity, thermal boundary layers, laminar and turbulent flows. Next, we explore forced internal heat transfer, beginning with a general material description of internal flow, average velocity, and average temperature. We proceed with the entrance region, overall heat analysis, plate and turbulent flow in pipes.

Finally, we discuss heat transfer enhancement techniques, which are divided into two types: active and passive methods and we conclude with the vortex generator.

I. 2 Heat transfer mechanisms :

Heat transfer is a field of science that allows us to estimate the rate of heat transfer or the amount of heat transferred per unit time. The latter occurs when there is a different temperature between two bodies. As we observe in daily life when we pour hot water into a cup, causing the temperature of the cup to increase and the temperature of the water to decrease over time, until the temperatures equalize. [3]

In conclusion, we can infer that there are different mechanisms of heat transfer, including conduction, convection, and radiation.

I. 2. 1 Conduction :

Conduction is the process of transferring energy from particles with higher energy to neighboring particles with lower energy in a solid body due to interactions between them [1]. In this case, thermal energy is transferred through the vibration of molecules. [3]

In this process, molecules gain kinetic energy from their original positions, and the range of vibration increases with higher levels of thermal energy.

I. 2. 2 Convection :

Convective heat transfer is defined as the mechanism of transferring heat within a fluid by the movement of the fluid from a hotter region to a cooler region. As such, the presence of liquid or gas motion is essential in facilitating the process of conducting heat transfer. [3]

There are three types of heat transfer: natural convection, forced convection and mixed convection. The rate of heat transfer in convection is calculated using Newton's cooling law according to the following equation: [3]

$$\emptyset = h \,.\, S \,.\, (T_s - T_f) \qquad \text{I.1}$$

h : convection heat transfer coefficient. $\left[\frac{W}{m^2 .K}\right]$

S : heat transfer surface area. $[m^2]$

T_s : Temperature of the surface. $[K]$

T_f : Temperature of the fluid sufficiently far from the surface. [K]

I.2.2.1 Natural Convection (free Convection) :

Heat is transferred through the movement of a fluid due to buoyancy forces. When a cold fluid is heated, a continuous flow called convection current occurs. This current is utilized in applications such as cooling electronic devices and designing efficient heat exchangers, in addition to understanding atmospheric phenomen like clouds and air patterns. [3]

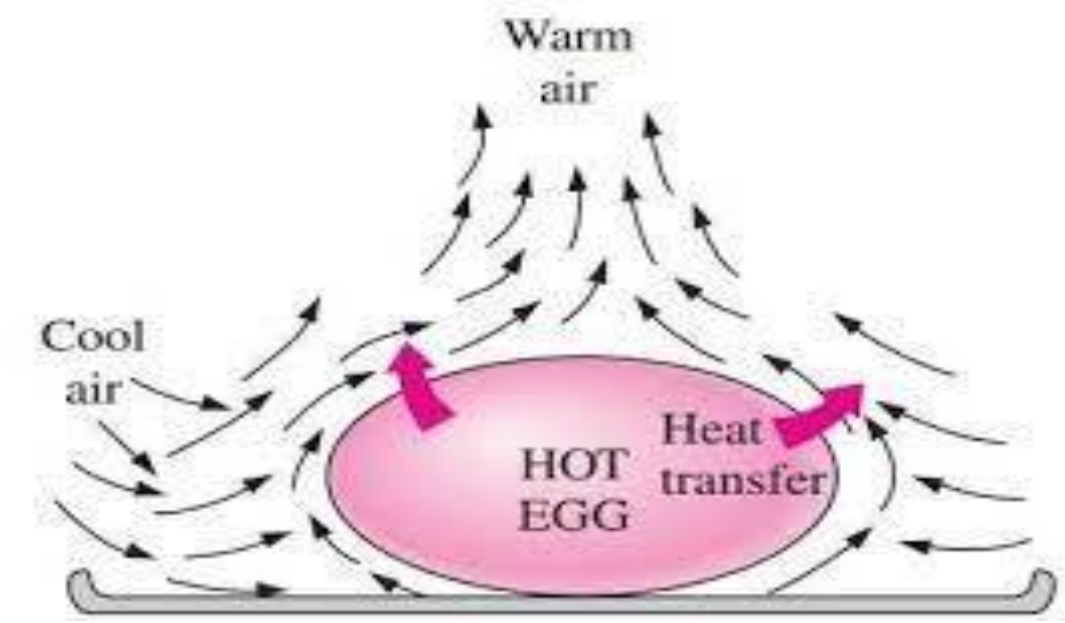


Figure I-1 : Natural convection.

I.2.2.2 Forced Convection :

Forced convection is a mechanism of transferring heat through the movement of fluid, where the fluid moves due to external pressure such as a pump or a fan. [1]

The fluid flows over a certain surface or inside a tube, allowing the efficient transfer of large amounts of heat energy. This mechanism is widely used in everyday life in central heating, air conditioning, steam turbines, and other machines.

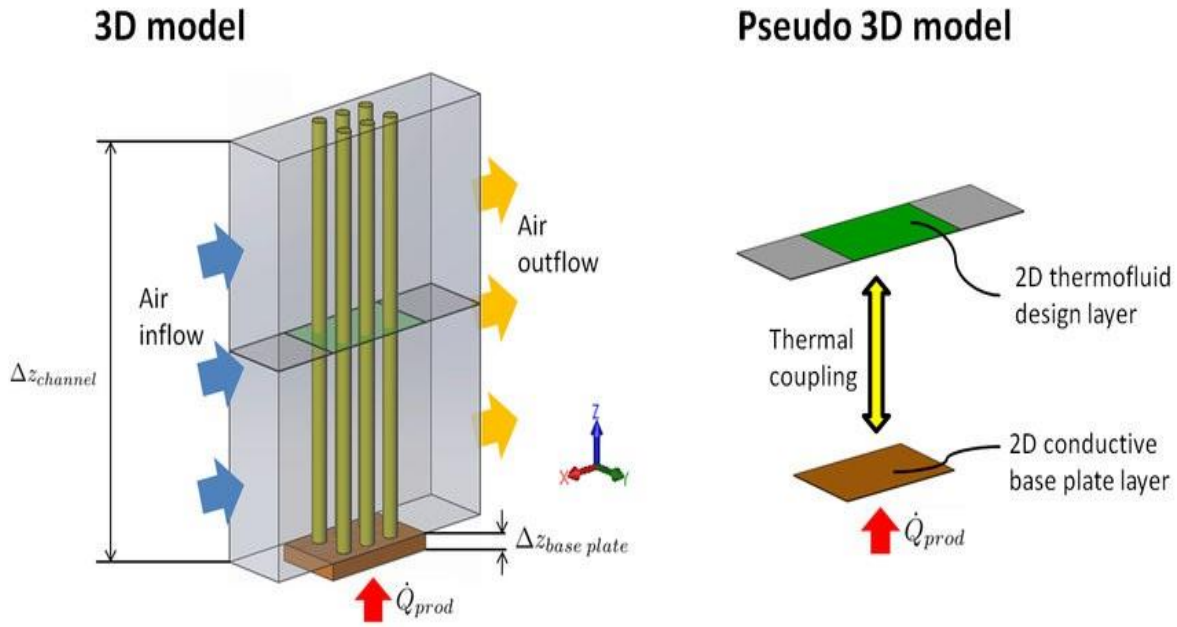


Figure I-2 : Forced convection.[13]

I.2.2.3 Mixed Convection :

In the context of forced convection, natural convective heat transfer is present to some extent unless the system is in a self-limiting or adiabatic state. When the influence of natural convective heat transfer is considerable, these flows are referred to as mixed convective heat transfer. Entropy plays a crucial role in heat transfer systems and fluid friction.

I. 2. 3 Radiation :

Heat transfer by radiation, also known as thermal radiation, is the process which heat is transferred through electromagnetic waves. It does not rely on any intermediate medium and can occur in a vacuum.

This type of heat transfer is governed by the laws of thermodynamics and involves radiation emission, absorption, and transmission of electromagnetic radiation.

Some common examples of heat transfer by radiation include the transfer of heat from the sun to the Earth, the emitted heat from a fire, or the radiant heat from a hot object. It is also an important factor in various industrial processes. [3]

I. 3 Dimensionless Number in Natural Convection :

Natural convection analysis in heat transfer, which is caused by fluid movement driven by temperature variations and density differences, involves the use of dimensionless numbers.

These dimensionless numbers help in understanding and predicting the behavior of natural convection. Some commonly used dimensionless numbers in the study of natural convection include:[6]

1. Grashof Number (Gr):

The Grashof number, denoted by Gr, is a dimensionless number used in the field of fluid dynamics to characterize the relationship between buoyancy forces and viscous forces in a fluid flow system.

It is defined as the product of the ratio of buoyancy forces to viscous forces and the ratio of characteristic length to the squared kinematic viscosity.

The formula for the Grashof number can be expressed as:

$$Gr = \frac{(g.\ \beta\ .\ \Delta T\ .\ L^3)}{\vartheta^2} \quad \text{I.2}$$

where:

g : is the acceleration due to gravity.

β : is the coefficient of thermal expansion.

ΔT : is the temperature difference across the fluid.

L : is the characteristic length representing dimensions of the system.

ϑ : is the kinematic viscosity of the fluid.

2. The Rayleigh number (Ra) :

Is a dimensionless number used in fluid dynamics, specifically in studies of buoyancy-driven flows, such as natural convection. It measures the relative importance of buoyancy forces compared to viscous forces in a fluid. The Rayleigh number (Ra) is defined as the product of the Grashof number (Gr) and the Prandtl number (Pr) :

$$Ra = Gr\ .\ \mathrm{Pr} \quad \text{I.3}$$

3. The Nusselt number (Nu)

The Nusselt number can be approximated using various correlations based on the physical conditions and geometry involved. One commonly used correlation for natural convection from a vertical plate is given by the following equation :

$$Nu = 0{,}54\ .\ Ra^{1/4} \quad \text{I.4}$$

Where : Nu : is the Nusselt number.

Ra : is the Rayleigh number, and the constants in the equation depend on the surface orientation and boundary conditions.

I. 4 Dimensionless Number in Forced Convection

Dimensionless numbers are used to quantify the relative importance of different parameters influencing heat transfer. Some of the important dimensionless numbers in forced convection are:[6]

1. Reynolds number (Re)

It relates the inertial forces to viscous forces and is given by the ratio of inertial forces (characterized by velocity) to viscous forces (characterized by viscosity).

It defined as :

$$\mathrm{Re} = \frac{\rho\ .U_0\ .D}{\mu} \quad \text{I.5}$$

Where : ρ : is the fluid density.

U_0 : is the velocity of the fluid.

D : is a characteristic diameter .

μ : is the dynamic viscosity of the fluid.

2. Prandtl number (Pr)

It represents the ratio of momentum diffusivity to thermal diffusivity of a fluid. The Prandtl number is essential in determining the velocity and thermal boundary layer thicknesses and hence influencing the convective heat transfer.[1]

It difined as : $\mathrm{Pr} = \frac{\mu . C_p}{\lambda}$ I.6

Where : μ : is the dynamic viscosity.

Cp : is the specific heat capacity at constant pressure.

ʎ : is the thermal conductivity.

The Prandtl numbers of fluids range from less than 0.01 for liquid metals to more than 100,000 for heavy oils (Table). Note that the Prandtl number is in the order of 10 for water .

Table I- 1: Typical ranges of Prandtl numbers for common fluids.

Fluid	Pr
Liquid metals	0.004–0.030
Gases	0.7–1.0
Water	1.7–13.7
Light organic fluids	5–50
Oils	50–100,000
Glycerin	2000–100,000

3. Nusslet number (Nu)

It describes the ratio of convective heat transfer to conductive heat transfer across a fluid boundary layer. The Nusselt number indicates the effectiveness of convective heat transfer and varies with different flow conditions. [6]

It is defined as : $\boldsymbol{Nu} = \frac{\boldsymbol{h.D}}{\lambda}$ I.7

Where :

h : is the convective heat transfer coefficient.

ʎ : is the thermal conductivity.

I. 5 Classification of fluid flow

Fluid mechanics is a branch of engineering and physics that deals with the study of fluid behavior in motion or at rest. Various engineering, kinematic, and physical standards are used in these sciences to describe and analyze the behavior of fluids during their evolution. In order to facilitate the study of fluid behavior and avoid the problems that researchers may encounter in fluid flow, fluids can be classified into groups based on their common characteristics.

T here are several general categories of fluids, including: [1]

I. 5. 1 Steady, Unsteady Flow

Steady flow is a term used to describe a situation where the properties of a fluid, such as dynamic viscosity μ, specific heat capacity C_p , thermal coductivity Λ and fluid density ρ , remain constant at any specific point within the flow over time.

On the other hand, unsteady flow is characterized by changes in these fluid properties over time. [1]

I. 5. 2 Laminar flow and Turbulent flow

Laminar and turbulent flow are two types of fluid motion. Laminar flow refers to a smooth and orderly movement of a fluid in distinct parallel layers, with minimal mixing between them.

On the other hand, turbulent flow is characterized by chaotic fluctuations and mixing of the fluid, resulting in irregular patterns and eddies. [2]

I. 5. 3 Compressible versous Incompressible Flow

Compressible flow refers to a flow where changes in fluid density are significant, typically occurring at high speeds or in gases.

Incompressible flow, on the other hand, assumes that the fluid density remains constant during the flow [1].

I. 5. 4 Viscous versus Inviscid Flow

When two fluid layers move relative to each other, there is a friction force that develops between them, causing the slower layer to resist the movement of the faster layer. This resistance is known as viscosity, which quantifies the internal stickiness of the fluid. Viscosity arises from cohesive forces between liquid molecules and molecular collisions in gases. Every fluid possesses some level of viscosity. Therefore, all fluid flows involve the effects of viscosity to some extent. Flows which exhibit significant viscosity effects are referred to as viscous flows. Conversely, idealized flows of fluids with zero viscosity are known as frictionless or inviscid flows. In these flows, the effects of viscosity are negligible, allowing for simpler analysis without sacrificing much accuracy.[2]

I. 5. 5 Natural flow versus forced flow

Natural flow, or unforced flow, refers to the movement of fluids that occurs through natural means such as buoyancy or gravity. In natural flow, no external force or energy is required to move the fluid. For example, in a solar water heating system, the warm fluid rises due to its lower density, while the cool fluid sinks due to its higher density. This effect allows for the circulation of the fluid without the need for a pump.

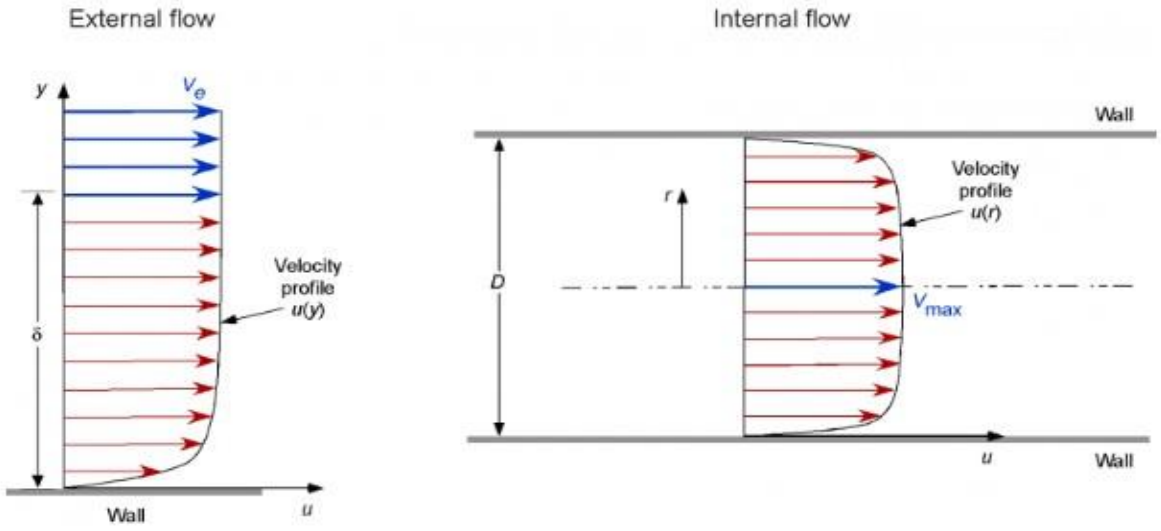
On the other hand, forced flow requires the use of external means such as pumps or fans to push the fluid over a surface or through pipes. In forced flow, the fluid is pushed or pulled by an external force, which provides the energy needed to overcome any resistance and move the fluid.

In general, the distinction between natural flow and forced flow lies in how the fluid's motion is initiated, where natural flow relies on natural forces and forced flow requires an external means to move the fluid.

I. 5. 6 Internal versus External flow

The classification of fluid flow as internal or external depends on whether the fluid is confined within a channel or flows over a surface. Internal flow occurs when the fluid is completely bounded by solid surfaces, such as in a pipe or duct. For example, the flow of water in a pipe is considered internal flow (Figure I-3-b).

On the other hand, external flow refers to the unbounded flow of fluid over a surface flowing. Additionally, when a pipe is partially filled with liquid and has a free surface, it is referred to as open-channel flow. This type of flow is seen in the flow of water in rivers and irrigation ditches (Figure I-3-a). [2]



(a) (b)

Figure I-3 : Internal versus External flow. [5]

I. 5. 7 One-Two and Three dimension flow

Fluid flow can vary in all three dimensions of a three-dimensional geometry, but some cases allow for simplification by treating the flow as one- or two-dimensional. For example, in a circular pipe, the velocity only varies in the radial direction, remaining constant in the angular and axial directions. However, the no-slip condition prevents uniformity across the pipe's cross-section, so the velocity is often assumed constant and uniform.[3]

I. 6 Velocity Boundry Layer

The flow of a liquid on a flat plate can be modeled using the surface of turbine blades and adjacent layers. The non-slip state causes particle velocity to zero, causing friction between layers. The plate's presence is felt up to the velocity limit layer, where sticky shear forces from fluid viscosity are felt. The flow is divided into boundary layers with significant sticky effects and speed changes, and a hidden flow area with minimal friction effects[1].as shown in the shape (Figure I-4) :

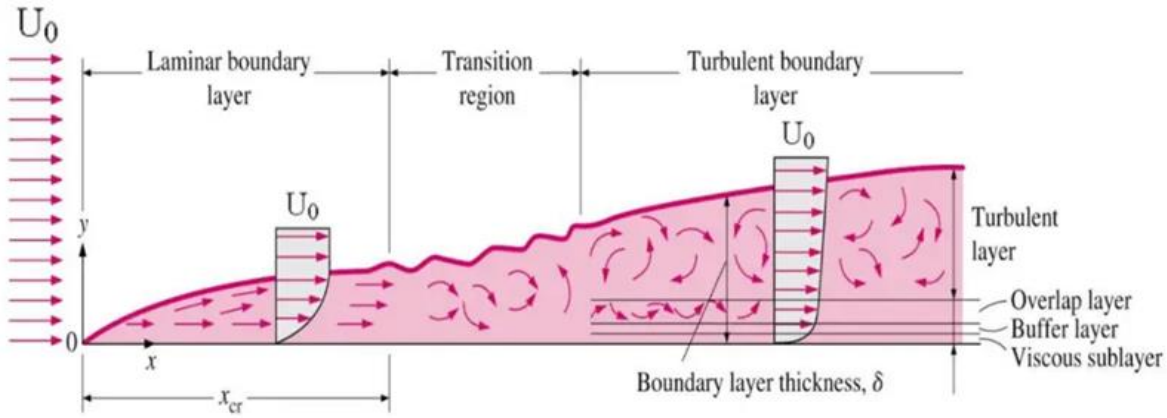


Figure I.4: Velocity Boundary Layer In Flat Plate.[1]

When we have a tube, a boundary layer forms for the velocity inside the fluid near the solid boundaries, and this layer is influenced by viscosity effects (Figure I-5). Viscosity is an important factor in determining flow behavior as it affects the movement of fluid molecules near the solid boundaries at a slower rate compared to fluid molecules in the outer part of the flow. The velocity inside the boundary layer differs due to this difference in movement. Simply put, the velocity boundary layer refers to the thin layer of fluid near the solid boundaries where changes in velocity and viscosity effects occur.

Understanding this phenomenon is crucial in studying flow behavior and fluid motion in its surroundings.[1]

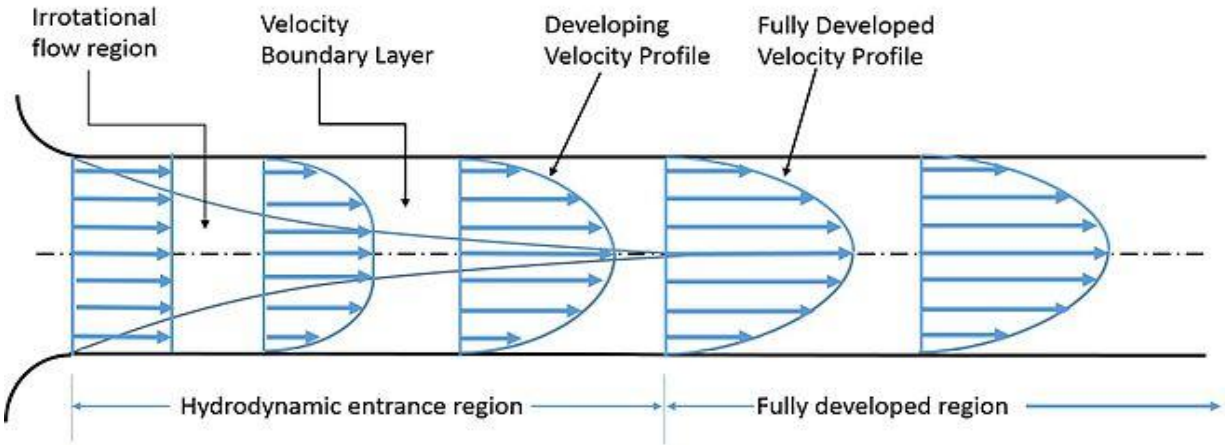


Figure I.5 : The development of the velocity boundary layer in a tube. (The developed mean velocity profile will be parabolic in laminar flow, as shown, but somewhat blunt in turbulent flow).

I. 7 Thermal Boundary Layer

The thermal boundary layer develops when a liquid flows at a certain temperature on a surface at a different temperature, as shown in the shape (Figure I-6): [1]

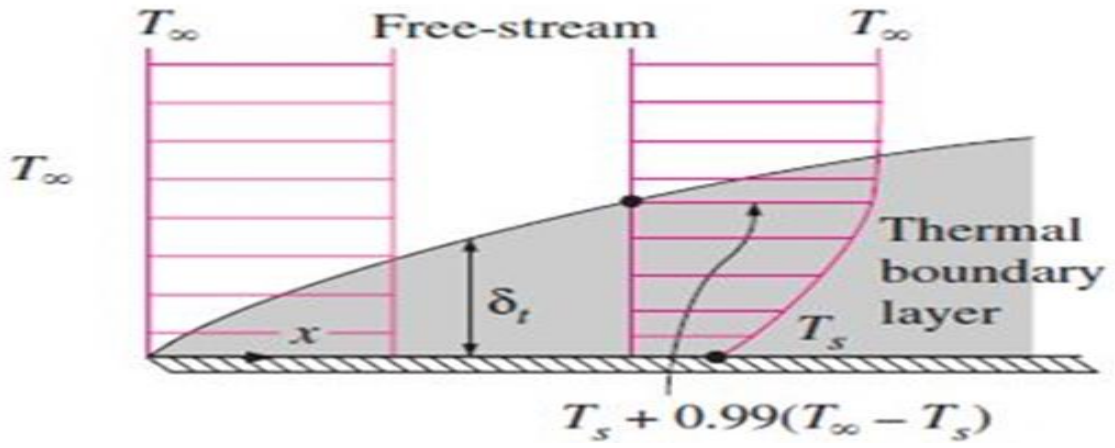


Figure I.6: Thermal Boundry Layer In Flat plate.

Thermal boundary layer: is a liquid flow area at T∞ temperatures above the surface where temperatures vary in a natural direction. The thickness of the boundary layer t is defined as the distance from the surface in which it is equal : $T - T_s = 0{,}99\ (T_\infty - T_s)$ along the surface. [1]

They are thickened in the direction of flow as the heat transfer effects between the solid surface and the flowing liquid are felt. Thermal boundary layer according to Prandlet is very thick in liquid metals and thin in oils. [11]

Heat transfer occurs when a fluid flows inside a tube, causing the wall layers to heat or cool. The fluid's core temperature is equal to the entry point'sT0, and does not participate in heat transfer.

However, temperature changes across wall layers create a thermal boundary layer near the tube surface, increasing in thickness with distance from the entry point. The entire fluid participates in heat transfer, resulting in dimensionless and similar temperature profiles. [1]

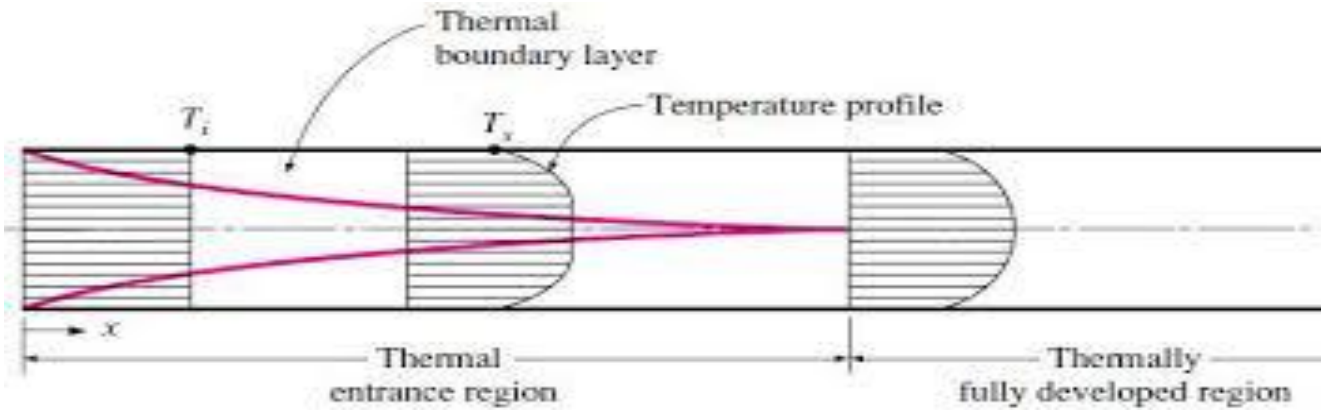


Figure I.7: Thermal Boundary Layer In Tube.

I. 8 Laminar flows and Turbulent flows

When a candle is lit, we can see that the smoke initially rises in a vertical direction for a few centimetres, and the flow pattern is smooth and orderly. However, after some time, it becomes turbulent and begins to fluctuate randomly in all directions while still rising (Figure I-8). [2]

This change in behavior indicates the transition of the flow from a smooth and orderly state (known as laminar flow) to a chaotic and unpredictable state (known as turbulent flow). This transition usually takes place in a specific region where the flow alternates between laminar and turbulent states, and it is termed transitional flow. [4]

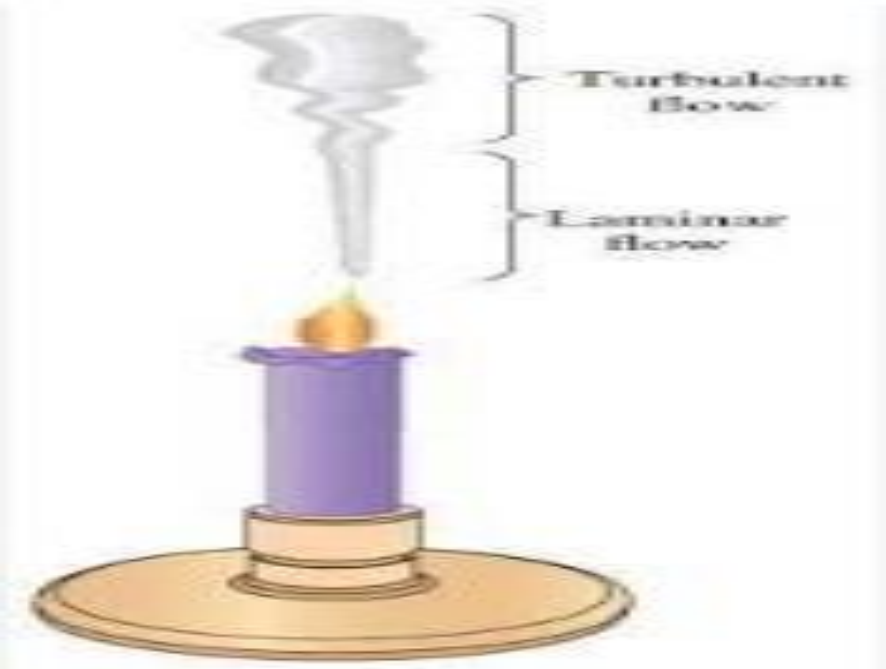


Figure I.8: Laminar and turbulent flow regimes of candle.[2]

The distinction between these three flow systems was experimentally established by the British engineer Osborne Reynolds more than a century ago. Reynolds conducted an experiment in which he injected dye into a glass tube.

By observing the bending and dispersion of the dye, he was able to detect fluctuations in the main flow and the rapid mixing of liquid molecules from adjacent layers. When the flow becomes completely turbulent, the friction factor reaches its maximum value. [1]

I. 9 Internal Forced Convection

In heating and cooling applications, focusing on the determination of friction factor and convection coefficient. It highlights the difference between exterior and interior fluxes, with external flow having a free surface and boundary layer expansion, and internal flow being entirely contained by tubes. The text covers mean temperature, velocity, and physical description of internal flow, as well as developing flow, fully developed flow, and hydrodynamic and thermal entry lengths. [7]

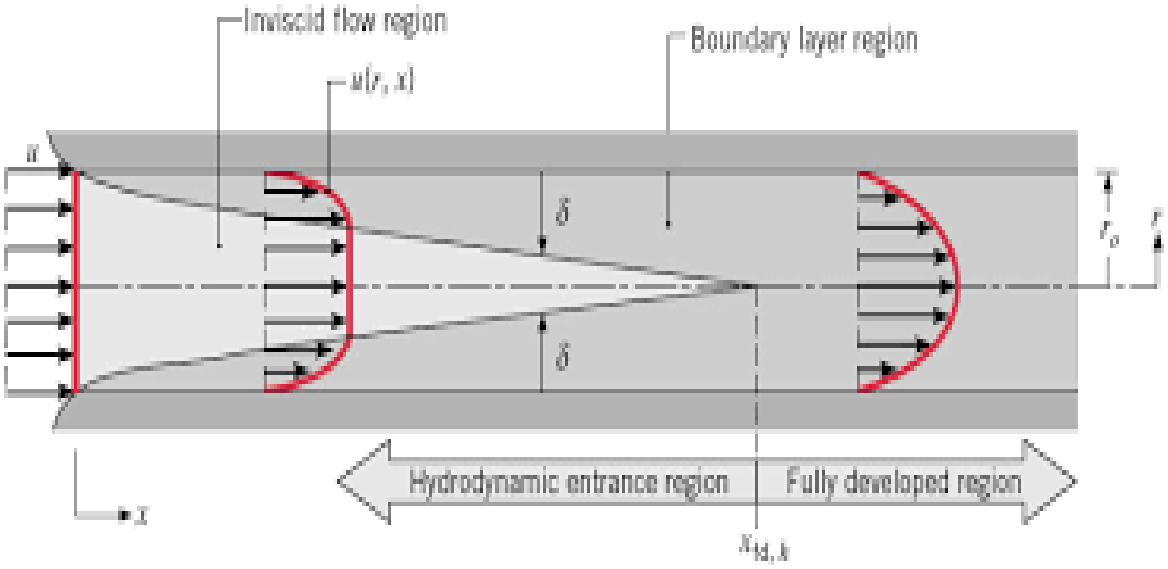


Figure I.9 : Internal forced convection. [12]

I. 9. 1 Average Velocity and Temperature

The free flow velocity is used to determine the Reynolds number and coefficient of friction, but there is no freedom of movement in inner flow. It is recommended to work with a medium speed V_m for incompressible flow velocities with a constant pipe section.

Average speed in heating and cooling applications may vary slightly due to temperature variations, but consistency in properties is usually more accurate. Mass conservation conditions determine average velocity V_m min pipes.[1]

$$\dot{m} = \rho V_m V_c = \int \rho V(x,y) dA_c \qquad \text{I.8}$$

Where : $\dot{m}$ is the mass flow, ρ is the density , A_c is the cross section and (r , x) is the velocity profile .

Then the average velocity of an incompressible flow in a circular pipe of radius R can be expressed as:

$$V_m = \frac{\int \rho V(x,y) dA_c}{\rho A_c} = \frac{\int_0^R \rho V(r,x) 2\pi dr}{\rho \pi R^2} = \frac{2}{R^2} \int_0^R V(r;x) r dr \qquad \text{I.9}$$

Average tempurature

When a liquid flows through a pipe, its temperature varies from wall surface to pipe center. It's advisable to maintain a uniform average temperature (T_m) throughout the route. The average temperature $T_m$is determined by the law of conservation of energy, ensuring the energy transferred equals the constant temperatur

$$\dot{E} = \dot{m}. C_p. T_m = \int C_p . T \delta \dot{m} = \int \rho. T. C_p. V dA_c \qquad \text{I.10}$$

The average temperature of a constant density, specific heat liquid in a circular tube of radius R can be calculated using the following formula:

$$T_m = \frac{\int C_p.T\partial\dot{m}}{C_P\dot{m}} = \frac{\int_0^R C_pT(\rho V2\pi r dr)}{\rho V_m(\pi R^2)C_p} = \frac{2}{V_m.R^2}\int_0^R T(r,x)\,V(r,x)dr \quad \text{I.11}$$

The average temperature $T_m$of a liquid changes with temperature change, and internal fluid properties are typically evaluated at the bulk medium's average temperature.such as :

$$T_b = (T_{m,i} + T_{m,e})/2 \quad \text{I.12}$$

I. 9. 2 The Entrance Region

In a circular tube, a fluid flows at constant velocity, creating a velocity gradient along the tube. The fluid particles in the contact layer with the pipe surface remain stationary due to friction, causing fluid particles in neighboring layers to slow down.

This results in a velocity gradient along the tube. The flow region is called the velocity boundary layer, where viscous shear forces act and flows become noticeable. The hydrodynamic layer entry zone divides flows through the pipe into two areas: the boundary layer area, where viscosity and velocity changes are significant, and the hydrodynamic entry length, where the flow in the inlet zone develops.[2]

I. 9. 3 The Fully Devoloped Region

Fully hydrodynamic developed regions remain unchanged, and shear stress on the pipe wall is related to the velocity gradient profile. The pressure drop in the hydrodynamic inlet region increases the average coefficient of friction, which can be significant for short tubes but negligible for long tubes.[2]

I. 9. 4 General Thermal Analysis

The energy conservation equation for a constant fluid flow in a pipe can be expressed as :

$$\dot{Q} = \dot{m}\,C_p(T_e - T_i) \quad \text{I.13}$$

Where : $T_i$and T_e : are average temperatures of the inlet and outlet fluid tubes.

Q̇ : is the temperature of the liquid circulating in the pipe.

The thermal conditions at the surface can be estimated with adequate accuracy at constant surface temperature (T_s=constant) or constant surface heat flux ($\dot{q_s}$ = constant).

The surface heat flow is expressed by :

$$\dot{q_s} = h_s(T_s - T_m) \quad \text{I.14}$$

If $h_s = h$ is constant, the surface temperature T_s must vary when $\dot{q_s}$ is constant, and the surface heat flux must vary when T_s is constant. Convective heat transfer is considered for both cases. [1]

I. 9. 5 Laminer flow in tube

The term refers to a substance that has no definite shape and takes the shape of its container, such as gases and liquids. When a fluid flows through a pipe, it experiences resistance caused by friction and viscosity.

If the average velocity of the fluid through the pipe is low, the fluid will flow or slide in parallel lines. In this case, the particles of the fluid move in an orderly manner and maintain their relative

positions on different sections during their motion. This type of flow is known as laminar or layered flow, and in this case Re < 2300.

For steady flow in a circular pipe with constant surface heat flux, the temperature distribution within the pipe can be obtained through the following formulation: [1]

$$\dot{q} = constant \qquad Nu = \frac{hD}{\lambda} = 4.36 \qquad \text{I.15}$$

For a constant temperature, the number of Nusselt can be calculated using the following equation :[1]

$$T = constant \qquad Nu = \frac{hD}{\lambda} = 3.66 \qquad \text{I.16}$$

The average Nusselt number for the thermal entrance region of a circular tube of length L subjected to a constant surface temperature can be determined by a equation : [1]

$$Nu = 3.66 + \frac{0.065(D/L)RePr}{1+0.04[(D/L)RePr]^{2/3}} \qquad \text{I.17}$$

It is used for significant variations in temperature leading to variations in dynamic viscosity of the fluid, it is also applicable for a condition of constant temperature at the wall, it is given by[1]:

$$Nu = 1.86\left(\frac{RePrD}{L}\right)^{1/3}\left(\frac{\mu_b}{\mu_s}\right)^{0.14} \qquad \text{I.18}$$

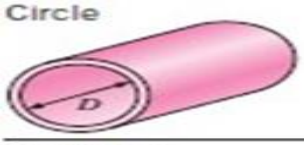
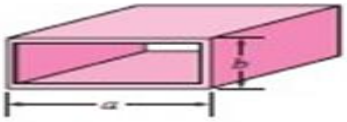
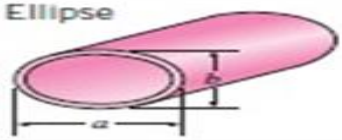
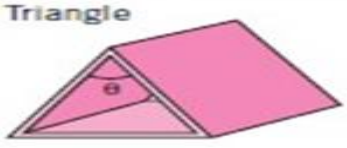
Tube Geometry	a/b or $\theta°$	Nusselt Number T_s = Const.	Nusselt Number $\dot{q}_s$ = Const.	Friction Factor f
Circle 	—	3.66	4.36	64.00/Re
Rectangle 	a/b			
	1	2.98	3.61	56.92/Re
	2	3.39	4.12	62.20/Re
	3	3.96	4.79	68.36/Re
	4	4.44	5.33	72.92/Re
	6	5.14	6.05	78.80/Re
	8	5.60	6.49	82.32/Re
	∞	7.54	8.24	96.00/Re
Ellipse 	a/b			
	1	3.66	4.36	64.00/Re
	2	3.74	4.56	67.28/Re
	4	3.79	4.88	72.96/Re
	8	3.72	5.09	76.60/Re
	16	3.65	5.18	78.16/Re
Triangle 	θ			
	10°	1.61	2.45	50.80/Re
	30°	2.26	2.91	52.28/Re
	60°	2.47	3.11	53.32/Re
	90°	2.34	2.98	52.60/Re
	120°	2.00	2.68	50.96/Re

Figure I.10: Nusselt number and friction factor for fully developed laminar flow in tubes of various cross sections ($D_h=4A_c/\rho$, $Re=V_mD_h/\nu$, and $Nu= hD_h/\lambda$).[1]

The average Nusselt number for the thermal entrance region of flow between isothermal parallel plates of length L can be expressed as follows: [1]

$$\text{Nu} = 7.54 + \frac{0.03(D_h/L)RePr}{1+0.016[(D_h/L)RePr]^{2/3}} \quad \text{I.19}$$

Where : $D_h$is the hydraulic diameter, which is twice the spacing of the plates.

This relation can be used for Re ≤ 2800.

I. 9. 6 Turbulent flow in tube

When a fluid flows through a pipe, it encounters resistance resulting from friction and viscosity. In the case of increased flow velocity, the fluid particles move in an irregular manner, creating swirling currents. This type of flow is called turbulent flow, and it has a Re >10,000.[1]

The Dittus-Boelter equation defined as :

$$Nu = 0.023Re^{0.8}Pr^{n} \qquad \begin{bmatrix} n = 0.4 \quad for\ heating \\ n = 0.3 \quad for\ cooling \end{bmatrix} \quad \text{I.20}$$

The Petukhov equation expressed as :

$$Nu = \frac{(f/8)RePr}{1.07+12.7(f/8)^{0.5}\left(Pr^{2/3}-1\right)} \qquad \begin{bmatrix} 0.5 \le Pr \le 2000 \\ 10^4 < Re < 5\times 10^6 \end{bmatrix} \quad \text{I.21}$$

$$f = (0.079lnRe - 1.64)^{-2} \qquad 10^4<\text{Re}<10^6 \quad \text{I.22}$$

The Gnielinski equation defined as :

$$\text{Nu} = \frac{(f/8)(Re-1000)Pr}{1+12.7(f/8)^{0.5}\left(Pr^{2/3}-1\right)} \qquad \begin{bmatrix} 0.5 \le Pr \le 2000 \\ 3\times 10^3 < Re < 5\times 10^6 \end{bmatrix} \quad \text{I.23}$$

$$f = \left(0.79\ln \text{Re}_D - 1.64\right)^{-2} \quad \text{I.24}$$

The relations above are not very sensitive to the thermal conditions at the tube surfaces and can be used for both T_S= constant and $\dot{q}_s =$ constant cases :

$$T_S = constant \quad ; \quad \text{Nu} = 4.8+0.0156Re^{0.85}Pr_S^{0.93} \quad \text{I.25}$$

$$\dot{q}_s = constant \quad ; \quad \text{Nu} = 6.3+0.0167Re^{0.85}Pr_S^{0.93} \quad \text{I.26}$$

I. 10 Heat transfer enhancement technique

Heat transfer enhancement methods can be classified into passive and active techniques. Active techniques use external energy to drive the process, while passive techniques do not require any additional energy to enhance the thermal and hydraulic performance in the system.

Passive techniques are widely used in experiments and numerical modeling to study heat transfer enhancement and reduce friction losses for energy and cost savings. Various passive techniques to

enhance heat transfer include the use of twisted wires, twisted or interlaced tapes, and vortex generator inserts.

This research focuses on evaluating the performance of twisted wires and twisted tapes as media for heat transfer enhancement, due to their ease of installation and low cost availability. The thermal performance of these components affects flow conditions such as laminar flow or turbulence. Through this current review, we will study investigations conducted on heat transfer enhancement using twisted wires and twisted tapes in laminar flow and turbulence regions. [8]

I. 10. 1 Passive method

Passive techniques are understood to involve changes in the geometry or surface of the flow channel to improve heat transfer efficiency in fluids. These techniques rely on the use of inserts, ribs, and roughened surfaces to increase fluid mixing and induce turbulence in the flow, ultimately leading to an overall enhancement in heat transfer. Negative techniques are characterized by their low cost, ease of installation, and production. [9]

I. 10. 2 Active method

Active techniques are more complex in design and application, as they require external power to control fluid flow and enhance thermal efficiency. It is challenging to provide external power in most practical applications, thus limiting the use of active techniques in practical fields. [10]

I. 11 Vortex Generator

One of the most effective techniques for enhancing thermal transfer is the incorporation of vortex generators, also known as chicanes, along the path of fluid flow. By introducing vortex generators of different shapes, the flow structure can be altered, leading to improved thermal transfer performance.

Placing a vortex generator near or away from the wall can completely change the flow direction and increase turbulence near the heat exchange surface. This increase in turbulence enhances the heat transfer process.

The significant role of vortex generators in heat transfer has prompted their widespread application in various industrial settings.[8]

I. 12 Conclusion

In conclusion, this chapter covered the fundamental concepts of heat transfer in tubes, starting with the three basic mechanisms : conduction, convection, and radiation.

It also reviewed different types of flow, including natural and forced convection, along with a detailed explanation of thermal and velocity boundary layers. The chapter highlighted heat transfer enhancement techniques, both active, requiring external energy, and passive, relying on changes in geometry or surface characteristics.

These concepts form the foundation for understanding and efficiently developing thermal systems, thereby improving the overall performance of engineering systems.

Chapter II :

Molding and Numerical Resolution Procedure

II. 1 Introduction

Commercially licensed software like Gambit and Fluent offer the capability to conduct 2D or 3D simulations in fluid mechanics. These simulations cover a broad spectrum, starting from mesh creation with Gambit to solving Navier-Stokes equations and post-processing with Fluent. Their robust graphical interfaces and extensive features support the simulation of diverse complex geometries, be it mobile or fixed, making them indispensable tools in industries like automotive, aeronautical, and aerospace, among others.

These software packages allow for the creation of fixed, relative, or adaptive meshes, and offer various physical models including multiphase and turbulent flow, enabling engineers and researchers to tackle a wide range of fluid dynamics problems.

II. 2 Conservation equation

Reservation equations are the basic principles used in physics and engineering sciences to describe mass, movement, and energy conservation laws. These models allow you to analyze physics systems by formulating mathematical equations that reflect the preservation of these quantities.

This comparison is important for understanding and analyzing the behavior of liquids, forces and natural phenomena, as well as for resolving these digital challenges and developing their application areas. [16]

II. 2. 1 Equation of quantity of motion

According to the Second Basic Dynamics Act, the change for movement of the liquid particle within a unit of time is equal to the amount of external forces operating on this particle [17]. It is expressed in Cartesian coordinates as follows:[16]

$$\sum_{j=1}^{2}\frac{\partial(\rho\overline{u}_i\overline{u}_j)}{\partial x_j}=\vec{F}-\frac{\partial p}{\partial x_i}+\sum_{j=1}^{2}\frac{\partial}{\partial x_j}\left[\mu\left(\frac{\partial\overline{u}_i}{\partial x_j}+\frac{\partial\overline{u}_j}{\partial x_i}-\frac{2}{3}\delta_{ij}\sum_{l=1}^{2}\frac{\partial\overline{u}_l}{\partial x_l}\right)\right]+\sum_{j=1}^{2}\frac{\partial}{\partial x_j}(-\rho\overline{u_i^{'}u_j^{'}}) \qquad \text{II.1}$$

The equation of the dynamics is written on both axes two- dimensional as follows :

Next x:

$$\left[\mathrm{u}\frac{\partial \mathrm{u}}{\partial x}+\upsilon\frac{\partial \mathrm{u}}{\partial y}\right]=-\frac{1}{\rho}\frac{\partial P}{\partial x}+\upsilon\left[\frac{\partial^2 \mathrm{u}}{\partial x^2}+\frac{\partial^2 \mathrm{u}}{\partial y^2}\right] \qquad \text{II.2}$$

Next y:

$$\left[\mathrm{u}\frac{\partial \mathrm{u}}{\partial x}+\upsilon\frac{\partial \mathrm{u}}{\partial y}\right]=-\frac{1}{\rho}\frac{\partial P}{\partial y}+\upsilon\left[\frac{\partial^2 \upsilon}{\partial x^{\acute{e}}}+\frac{\partial^2 \upsilon}{\partial y^2}\right]+\rho g\beta(T-T_0) \qquad \text{II.3}$$

With : fluid incompressible : $\rho = cts$

Fluid stasionnair : $\frac{\partial \rho}{\partial t}=0$

II. 2. 2 Continuity equation

It expresses the principle of mass retention for a non-compressable neutron liquid as well as for material control volume, given as follows:[8]

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_i}\left(\rho \overline{U_i}\right) = 0 \qquad \text{II.4}$$

In fixed case, the derivative is cancelled relative to the time, then the block save is written (fluid stasionnaire : $\frac{\partial \rho}{\partial t} = 0$) :

$$\frac{\partial}{\partial x_i}\left(\rho \overline{U_i}\right) = 0 \qquad \text{II.5}$$

If the density of the material is constant (fluid incompressible $\rho = cts$), we get :

$$\frac{\partial}{\partial x_i}\left(\overline{U_i}\right) = 0 \qquad \text{II.6}$$

With : $\rho = 0$

Fluid two-dimensional:

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0 \qquad \text{II.7}$$

II. 2. 3 Energy equation

This is obtained through the application of the first heat principle and the principle of energy conservation.[8]

$$[u\frac{\partial T}{\partial x} + v\frac{\partial T}{\partial y}] = \lambda\,[\frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2}] \qquad \text{II.8}$$

Equation after application of conditions:

Material : stationary: $\frac{\partial \rho}{\partial t} = 0$

Incompressible : ρ = cts

two-dimensional : (x,y)

II. 3 Presentation of the GAMBIT Software

Gambit software (Geometry And Mesh Building Intelligent Toolkit) is a 2D/3D meshing tool ; a preprocessor that allows meshing geometrical domains of a CFD problem. It enables the generation of structured or unstructured eshes in Cartesian, polar, cylindrical, or axisymmetric coordinates.

It can create complex meshes in two or three dimensions with mesh types like rectangles or triangles. The mesh generation options in Gambit offer flexibility of choice. Users can decompose the geometry into multiple parts to generate a structured mesh. Alternatively, Gambit automatically generates an unstructured mesh suited to the constructed geometry type. With mesh verification tools, defects are easily detected.

Gambit can be used to construct geometry and generate a mesh for it, or geometry from another CAD software can be imported into this preprocessor. It generates *.msh files for Fluent. [14]

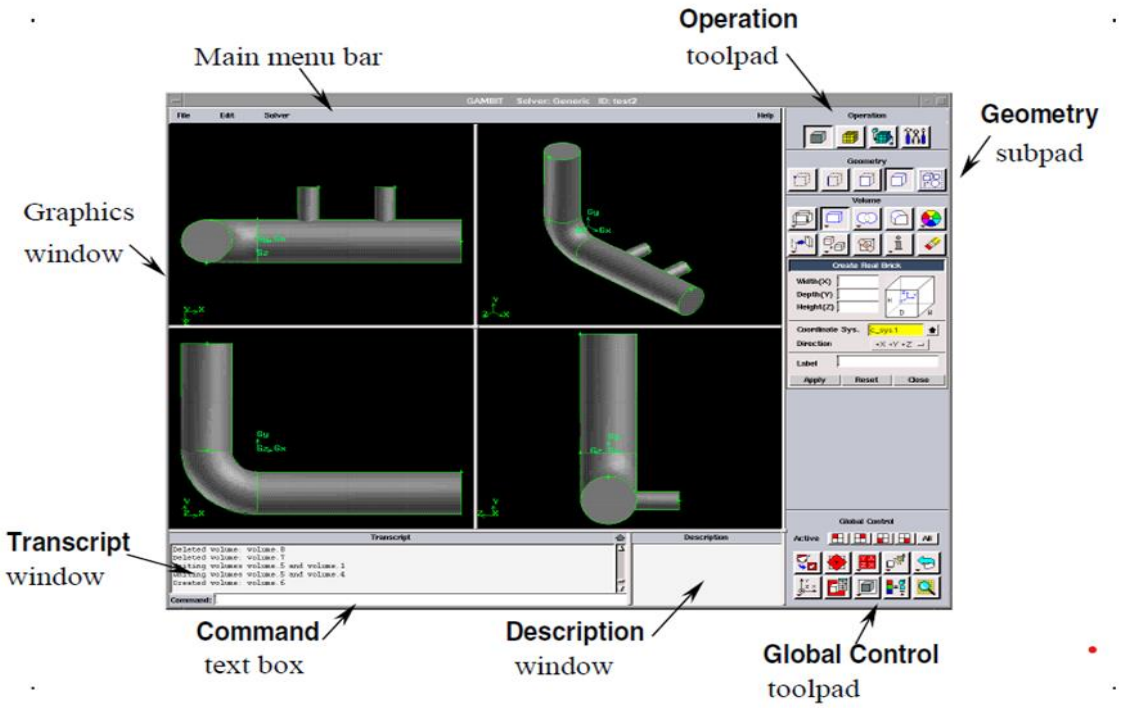


Figure II.1: The gambit guide.[18]

II. 3. 1 Starting GAMBIT

The Gambit application path is as follows :

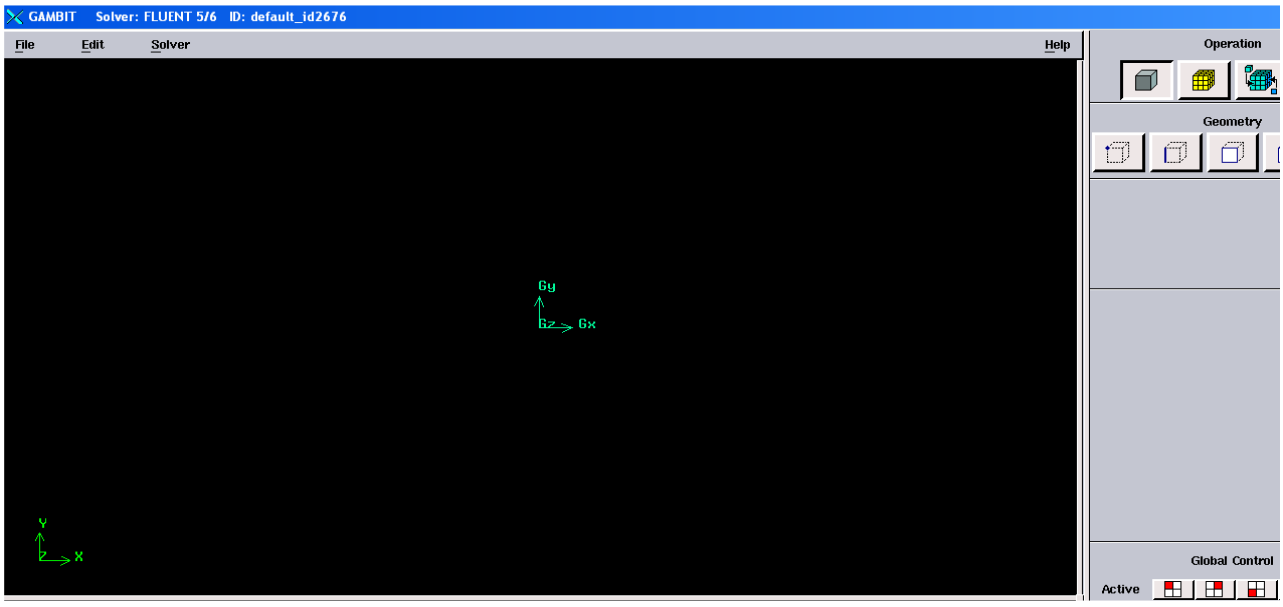


Figure II.2: Launch of the gambit.[14]

II. 3. 2 GAMBIT Features

Gambit combines three main functions: defining problem geometry (constructing if the geometry is simple or importing CAD geometry), meshing and verification, and defining boundaries (types of boundary conditions) and calculation domains. If the geometry is designed using CAD software like SolidWorks or CATIA, it is advisable to import files in ACIS format (*.sat), and often it is necessary to clean up the geometry.

In the top right corner, you will find the various operations that can be performed in this software, as illustrated. [14]

- The first operation is geometry. This menu allows creating volumes, surfaces, lines, and points. It also enables the modification of imported geometries.
- The second operation is geometry meshing. This menu facilitates meshing volumes, surfaces, and turning lines into points.
- The third operation allows for defining boundary conditions and fluid domains.
- The fourth button is a tool menu.

By clicking on each functionality of the main menu, a corresponding secondary menu will appear just below.

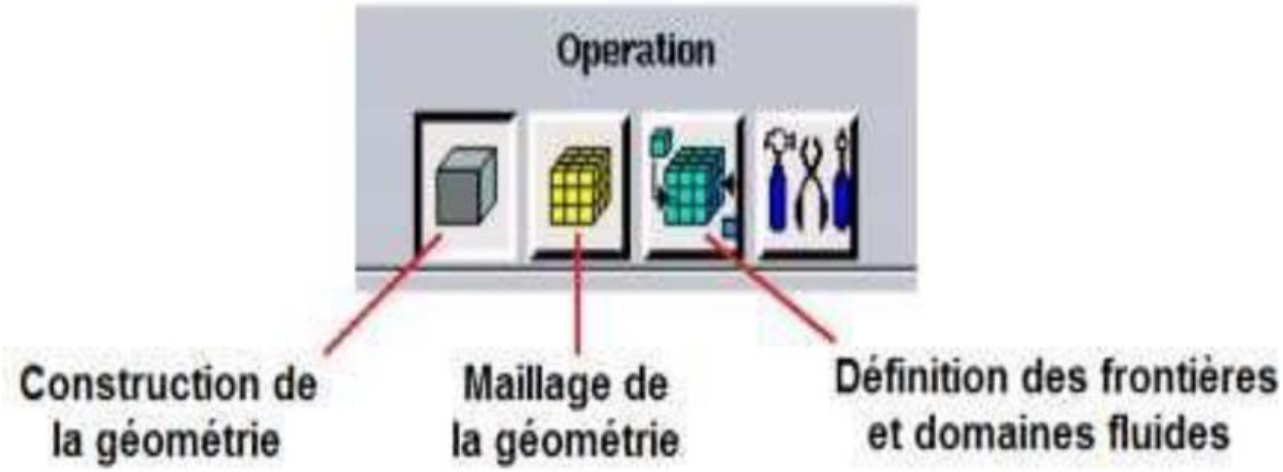


Figure II.3 : Main functions of the Gambit general menu.

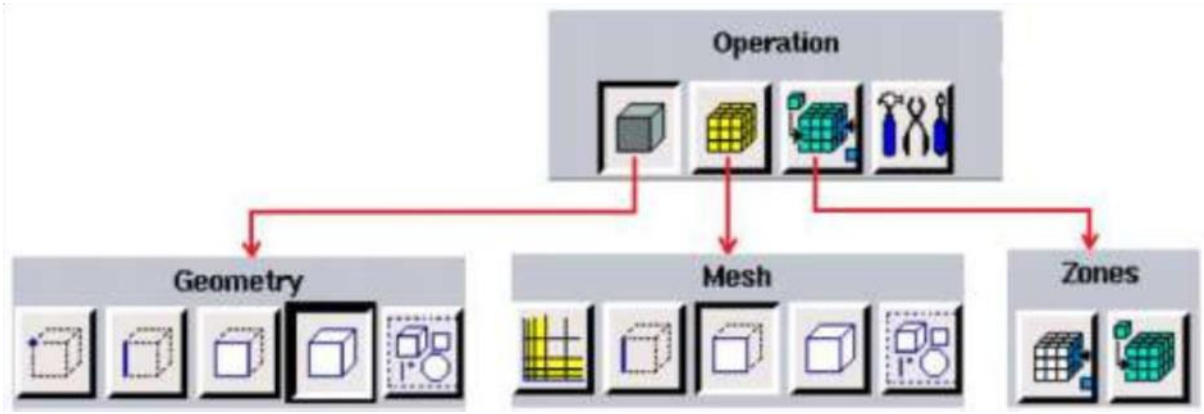


Figure II.4: Secondary Menus of Gambit.

II. 3. 3 Description of Small Geometry in Gambit

The purpose of constructing geometry is to define the calculation domains that will be surfaces in a 2D problem and volumes in a 3D problem. [14]

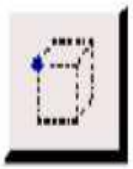
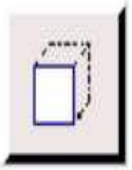
Commande	Point	Segment	Face	Volume	Groupe
Symbole					

Figure II.5: Table Description of commands in the geometry menu.

II. 3. 4 Description of the mesh menu

The mesh menu consists of five buttons as shown in the table below. [14]

Commande	Couches limites	Segment	Faces	Volume	Groupe
Symbole					

Figure II.6 : Table Description of commands in the mesh menu.

The description of the main mesh menus, as shown in Figure II.7, is as follows:

1. Creation and modification of a boundary layer mesh.
2. Size of the first cell.
3. Mesh expansion factor.
4. Number of lines of the specific mesh.
5. Mesh type (4 specific mesh types are available).
6. Line meshing.
7. Application of a double ratio that increases point density either on the sides or at the center of the lines.
8. Use of a ratio for meshing.
9. Choice of mesh option: number of nodes or interval size between the two.
10. Face meshing.
11. Smoothing of deformed meshes.
12. Choice of mesh type: Tri, Quad.

13. If option checked, creation of a regular mesh according to the parameters below. If not, creation of a mesh from nodes defined on the edges.
14. Volume meshing.
15. Smoothing of deformed meshes.
16. Choice of mesh type: Tetra, Hex.
17. If option checked, creation of a regular mesh according to the parameters below. If not, creation of a mesh from nodes defined on the edges.

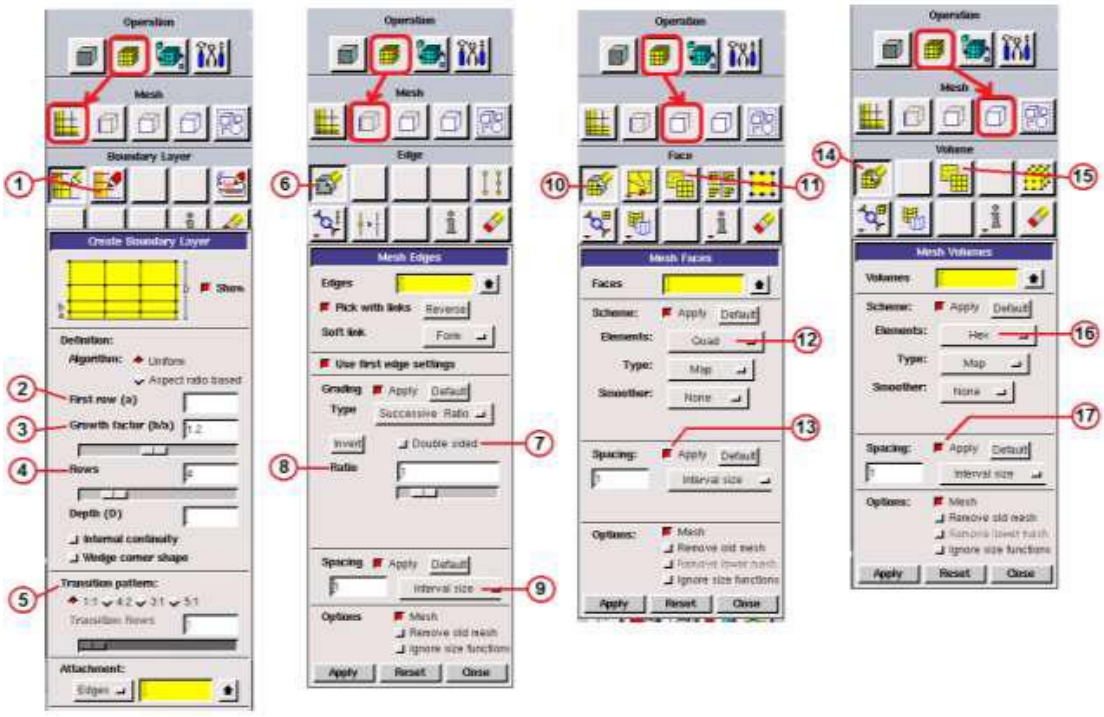


Figure II.7: Main mesh menus in Gambit.

II. 3. 5 Boundary conditions and definition of domains

Several types of boundary conditions are available depending on the problem being addressed. It is important to note that the meshed space is assumed to be fluid by default. The name assigned to each boundary is crucial, as it will be carried over to Fluent. Unclear names may lead to boundary confusion.

The Gambit meshing tool can generate meshes that many solvers can utilize. Therefore, we need to specify the solver software with which we want to process the mesh file. The choice of boundary types varies depending on the solver considered for solving the problem.

Common types of boundary conditions include: [14]

- Velocity inlet: Used for incompressible or moderately compressible flows when the inlet velocity is known.
- Pressure Inlet: Used for compressible and incompressible flows.

- Mass Flow Inlet: Imposes a known mass flow rate at the inlet. Mass Flow Inlet is not necessary for incompressible flow.
- Pressure Outlet: This type is used to define the static pressure at the outlet. Using "Pressure Outlet" instead of "Outflow" often results in better convergence.
- Outflow: This condition is used to model fluid outlets where the details of velocity and pressure at the outlet are not known in advance. It is not appropriate for the following calculations.
- If the problem has a "Pressure inlet" condition.
- If you are modeling compressible flow.
- If you are modeling unsteady flow with density variation.
- Wall: This type is used to delineate solid regions from fluid regions. Smooth wall properties are generally used.

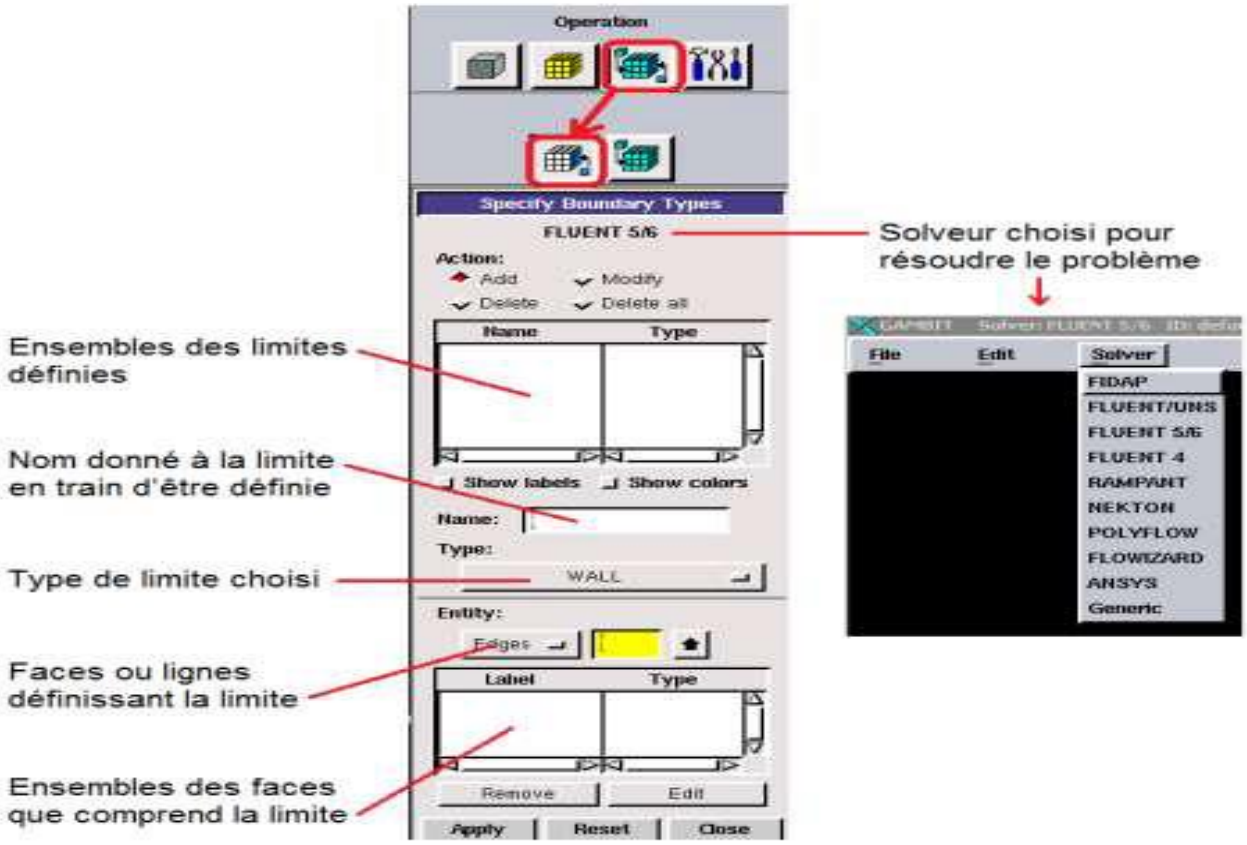


Figure II.8 :Main menus for boundary conditions.

II. 3. 6 Display Menu

The global control window concerns orientation, display, lighting, and mesh checkin (Fig II.9). Just to the right of the word 'Active,' there are five icons that are used to activate different views. [14]

Figure II.9 : Gambit Display Menus.

The rest of the icons have the following functionalities:

This display option adjusts the zoom to fit the dimensions of the window.

Allows rotation relative to the origin of the local coordinate system.

Visualize the three views projected onto the global coordinate system and an isometric view.

Allows modifying the 'lighting' of the room and adding annotations.

Go back or forward.

Show the view according to the global reference frame.

Allows choosing the entities or meshes to make visible.

View with or without hidden edges, renders the faces solid.

Change the colors to distinguish different entities.

Allows examining the mesh plane by plane.

II. 3. 7 Export of the gambit mesh

Once the geometry is created and the boundary conditions are defined, the mesh should be exported in. mesh format so that Fluent can read and use it. Afterwards, Gambit can be closed, saving the session if desired, and Fluent can be launched.

So, once the geometry is created and boundary conditions are defined, the mesh is exported as a file (mesh: meshing) for easy reading by Fluent software. [14]

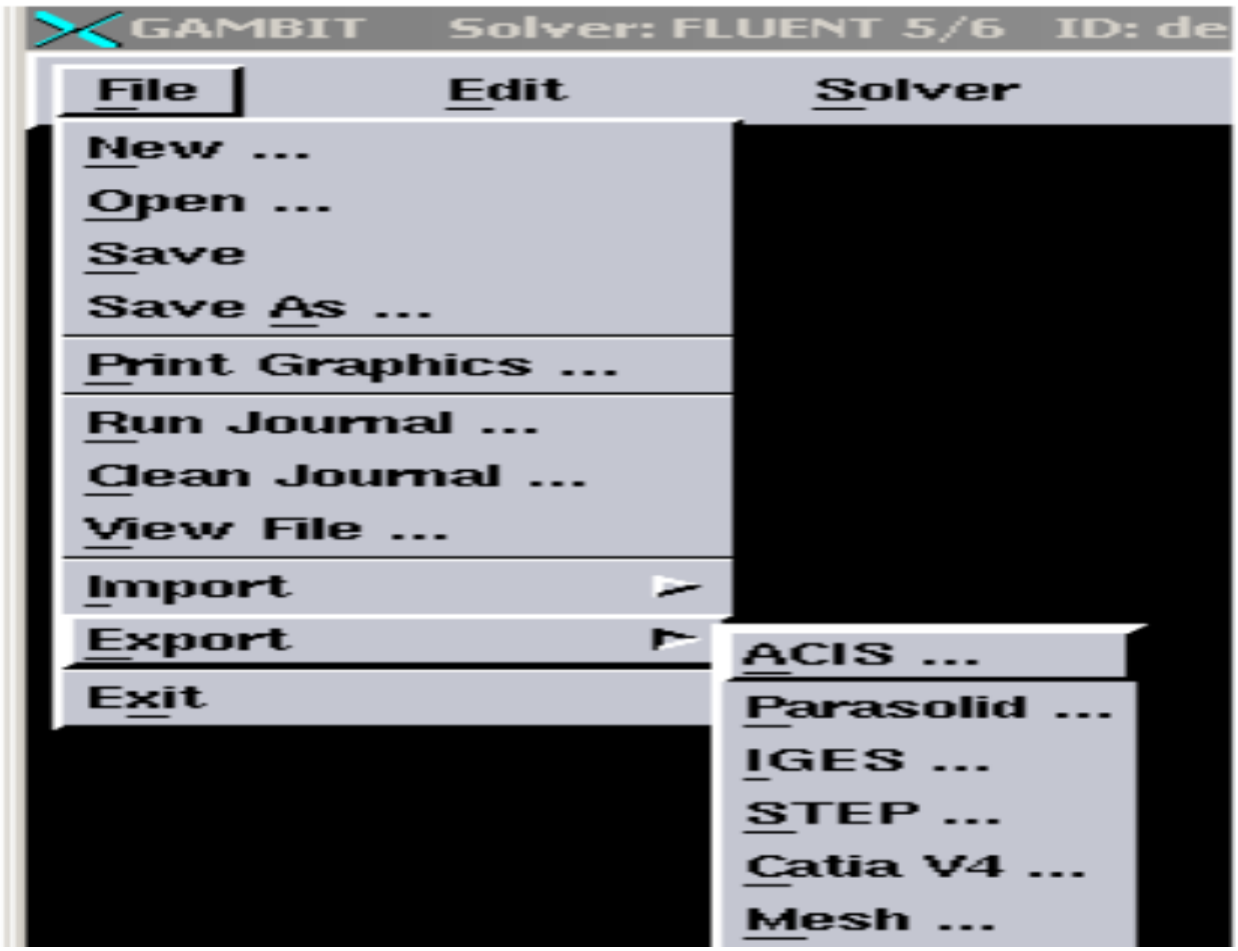


Figure II.10: Exporting the mesh from Gambit.

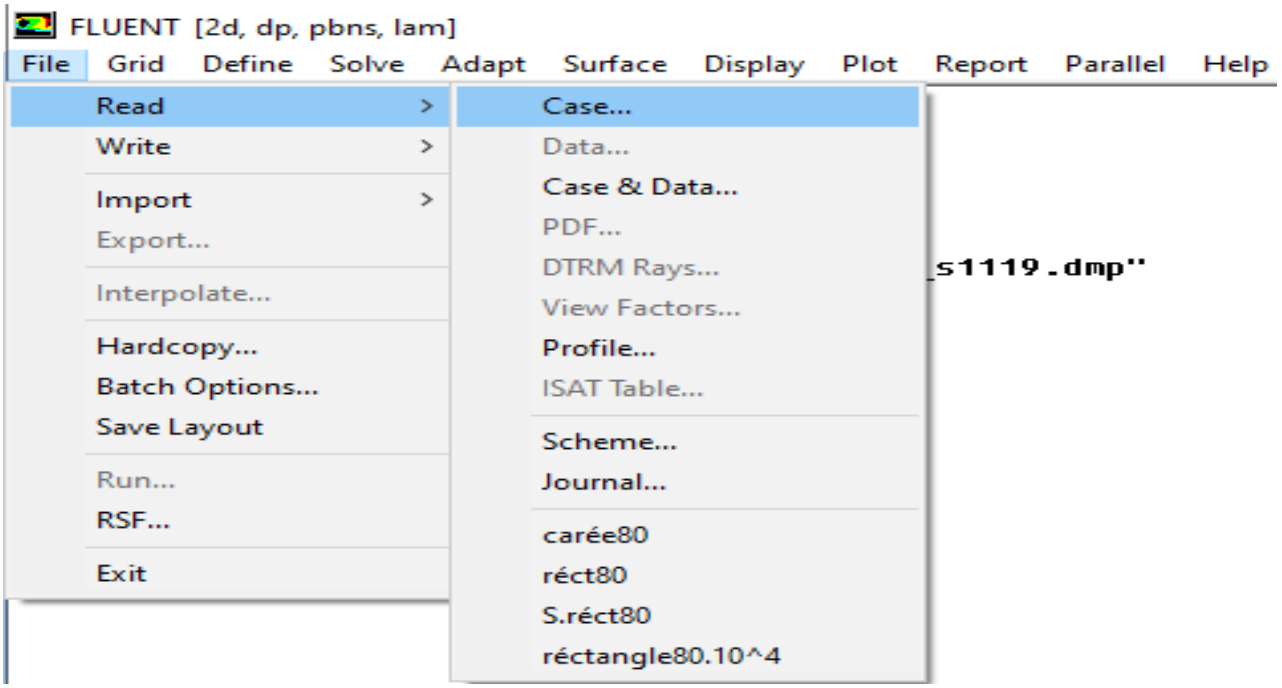
II. 4 Fluent software

Fluent is a powerful software for digital fluid simulation in industries, enabling the modeling and simulation of fluid behavior in various systems. It is essential for applications in aerospace, automobiles, energy, and product design optimization.

Fluent is widely used software in the industry to solve complex fluid problems. It enables engineers and researchers to simulate and analyze fluid flows in various systems, like airplanes, cars, heat exchangers, turbines, and reactors.

Fluent optimizes product design, reduces development costs, improves performance, and anticipates potential problems before production. It also reduces costly physical tests and saves time.

Fluent is ANSYS software for digital fluid simulation, allowing the modeling and resolution of complex mathematical equations describing fluid movements, including laminar and turbulent flows, thermal convection, combustion, chemical reactions, and multiphasic reactions.

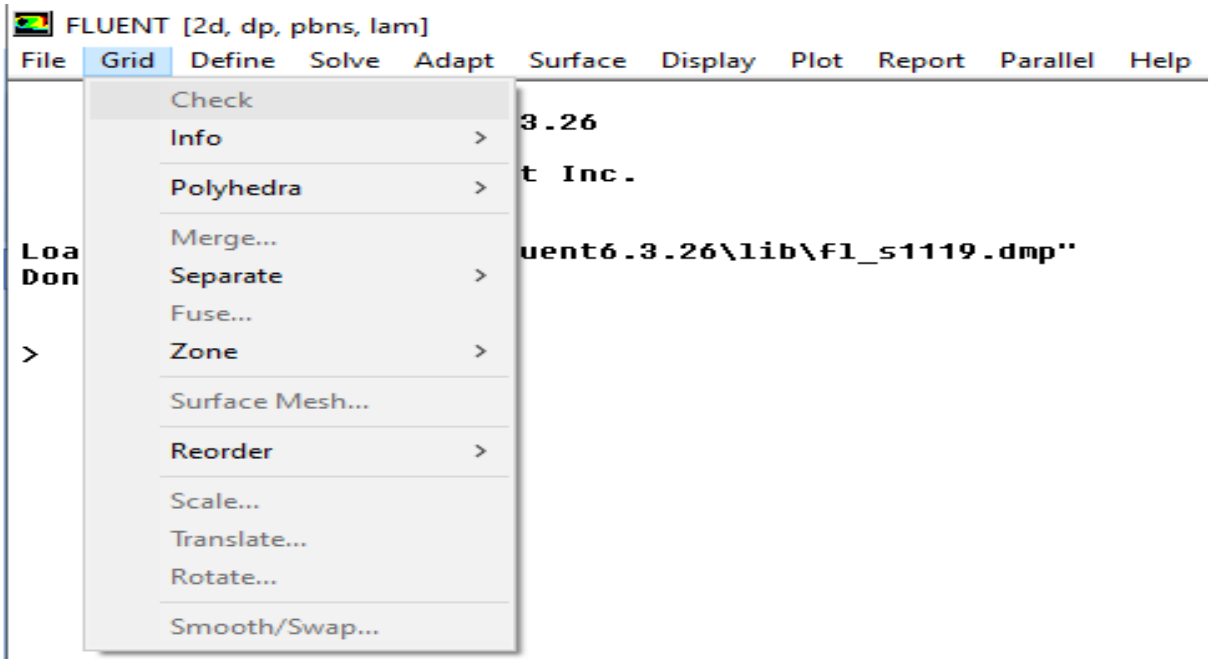


Figuer II.11: To start the simulation you need to import the file. [14]

II. 4. 1 Verification of the mesh matters

The aim of this work is to evaluate the quality of the flux-imported maillage, identify potential issues or errors, and ensure it is geometrically conforming to the model's geometry for precise and realistic simulation.

This step allows you to check whether the imported mesh contains no errors or negative volumes. [18]

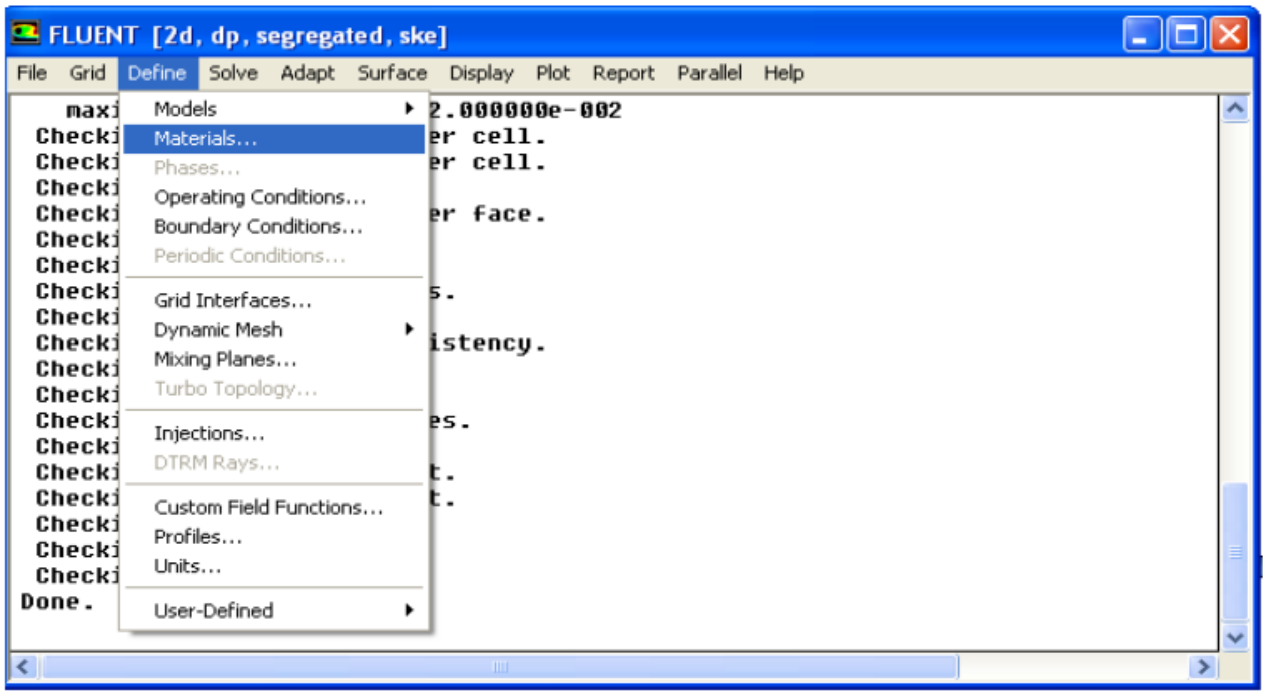


Figuer II.12: Verification of Fluent sou mesh.

II. 4. 2 Definition of fluid characteristics

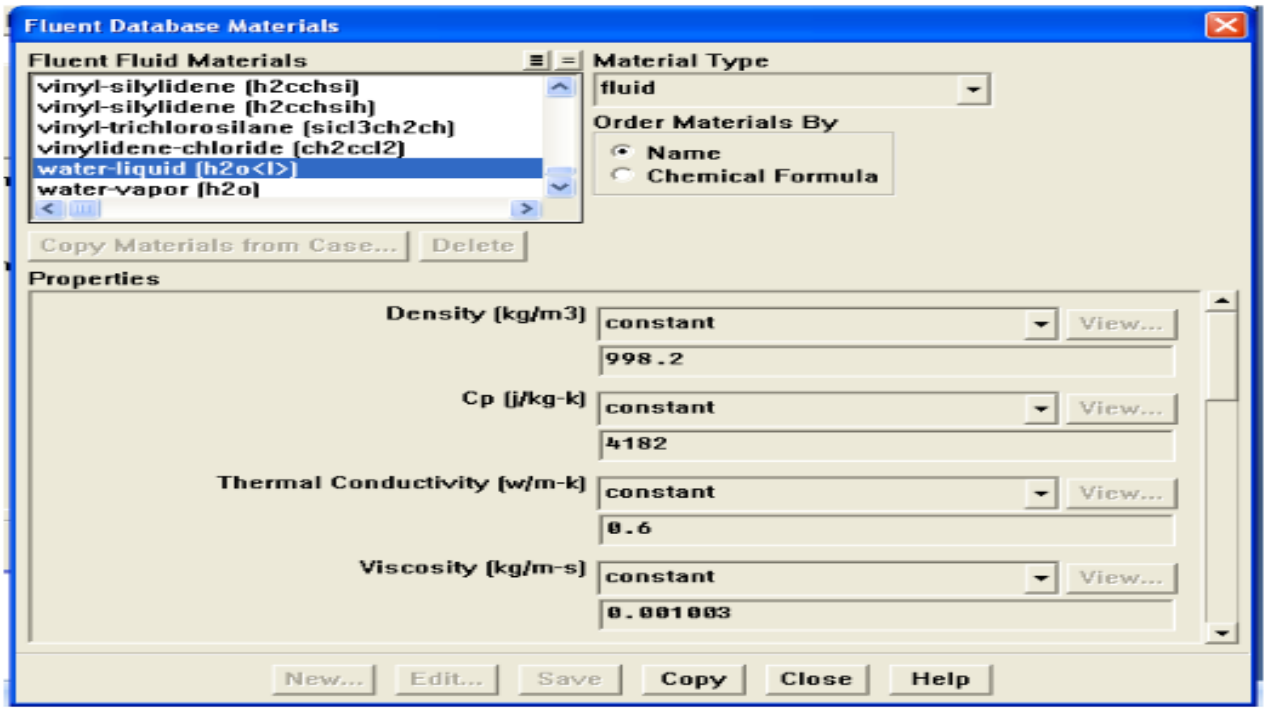
The Fluent Database is used to download and verify known liquid properties. [15]

Define / Materials



Figuer II.13: Definition of fluid characteristics.

The pressure on « copy » shows the next interface.



Figuer II.14 :Fluid List Interface.

Solide (copper)

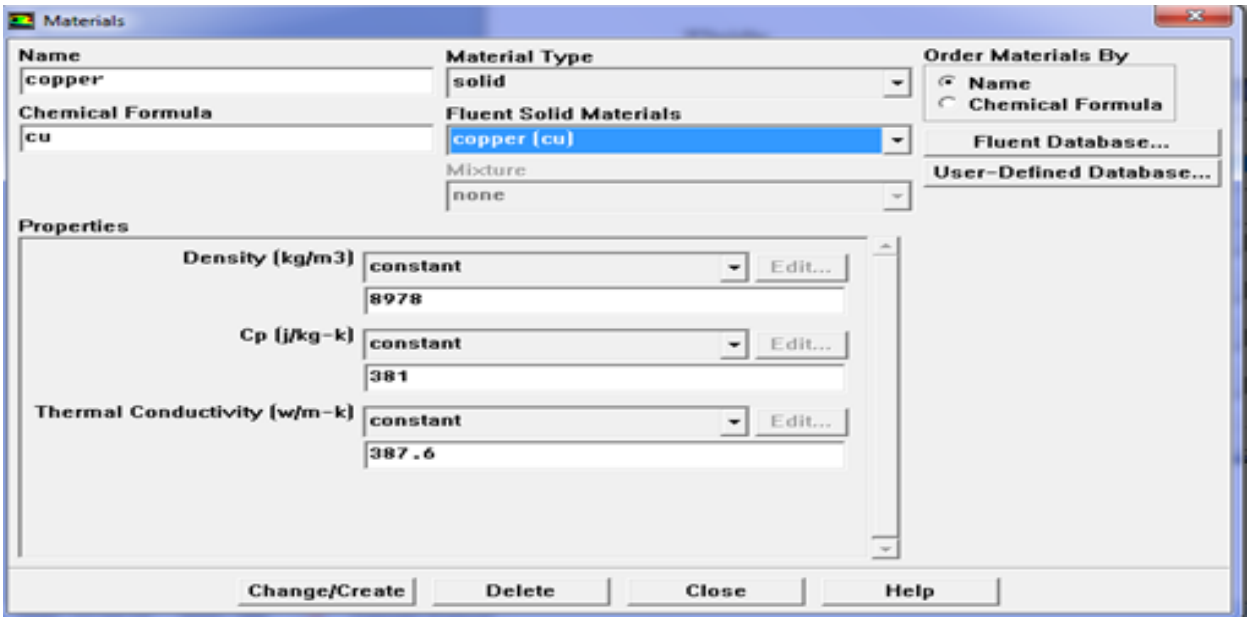
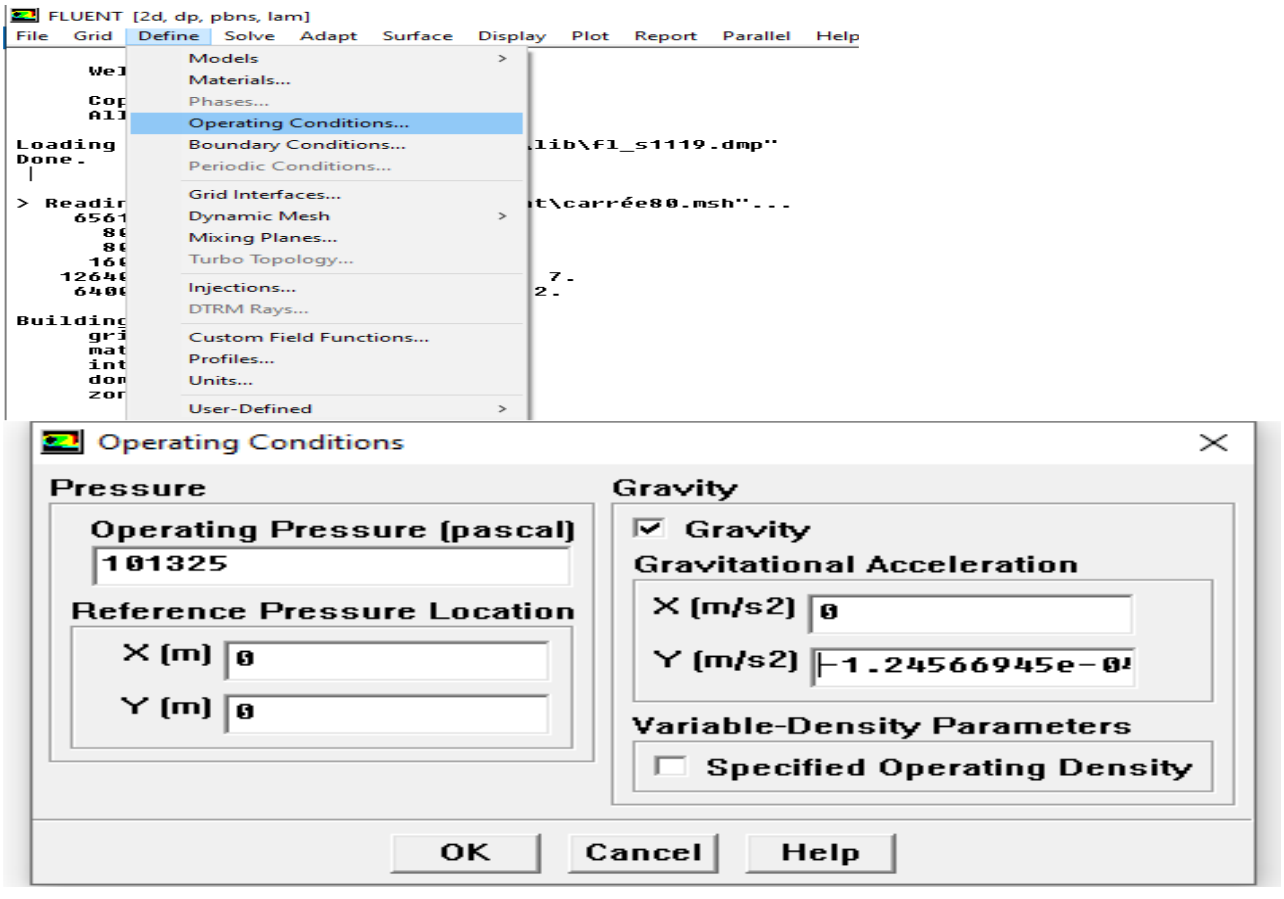


Figure II.15: Solid List Interface.

II. 4. 3 Introduction of gravity

By putting pressure on operating conditioning, selecting gravity, and inputting the value required. [14]



Figuer II.16: Introduction of gravity.

II. 4. 4 Condition at limit

Define / Boundary Conditions

The values of the physical limits entered into the gambit must be set and given at the entry and exit of the liquid. We adjust the velocity and the pressure, and for the wall, we can adjust the flow, the temperature, and so on. [22]

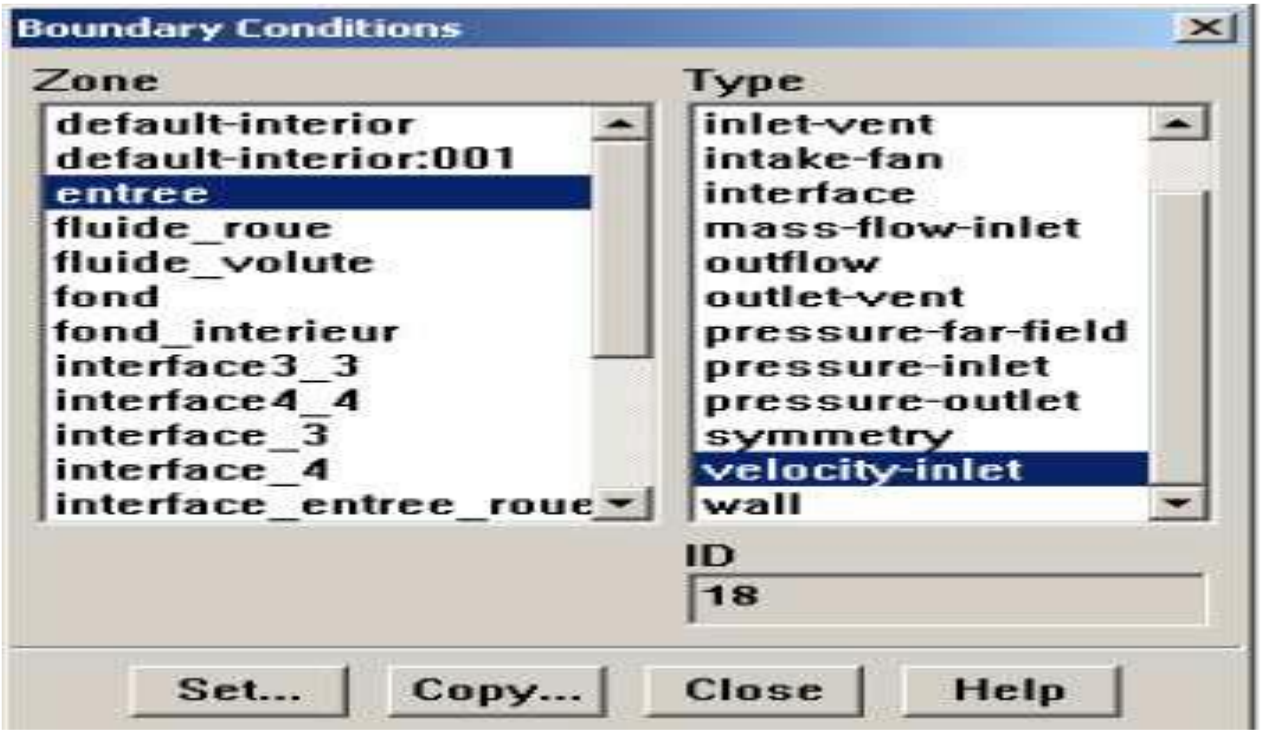
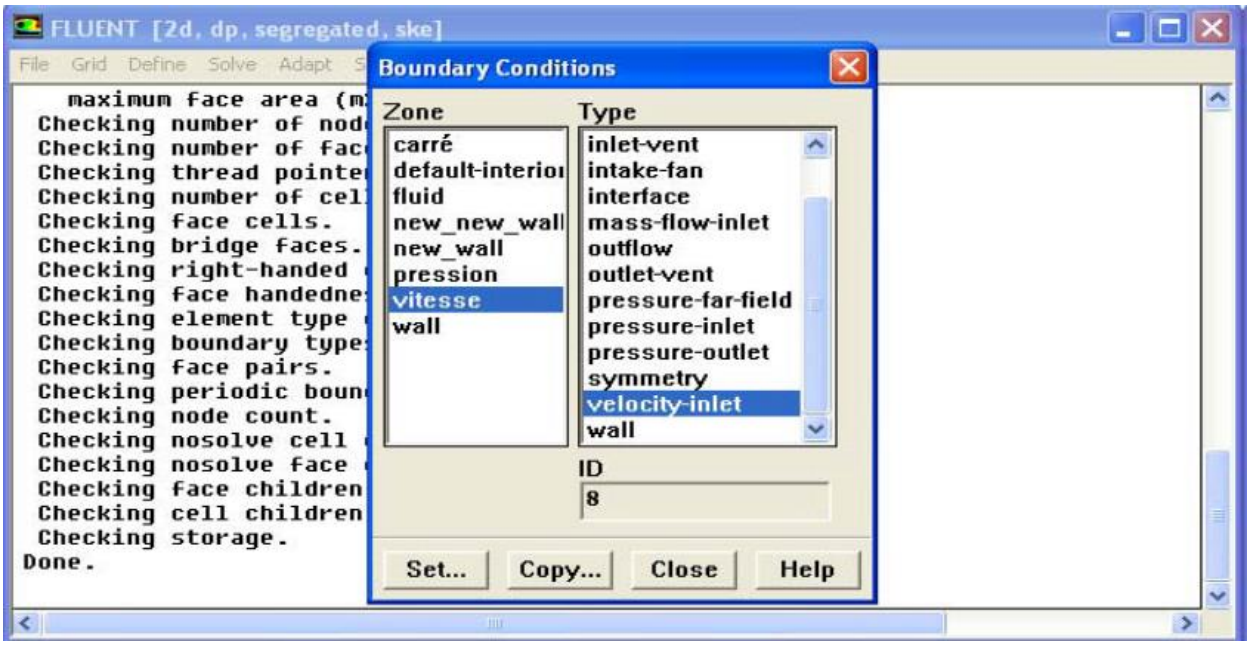


Figure II.17 : Limit Condition Values.

- **Velocity inlet**



FigureII.18 : Input Speed (velocity inlet).

- **Pessure inlet**

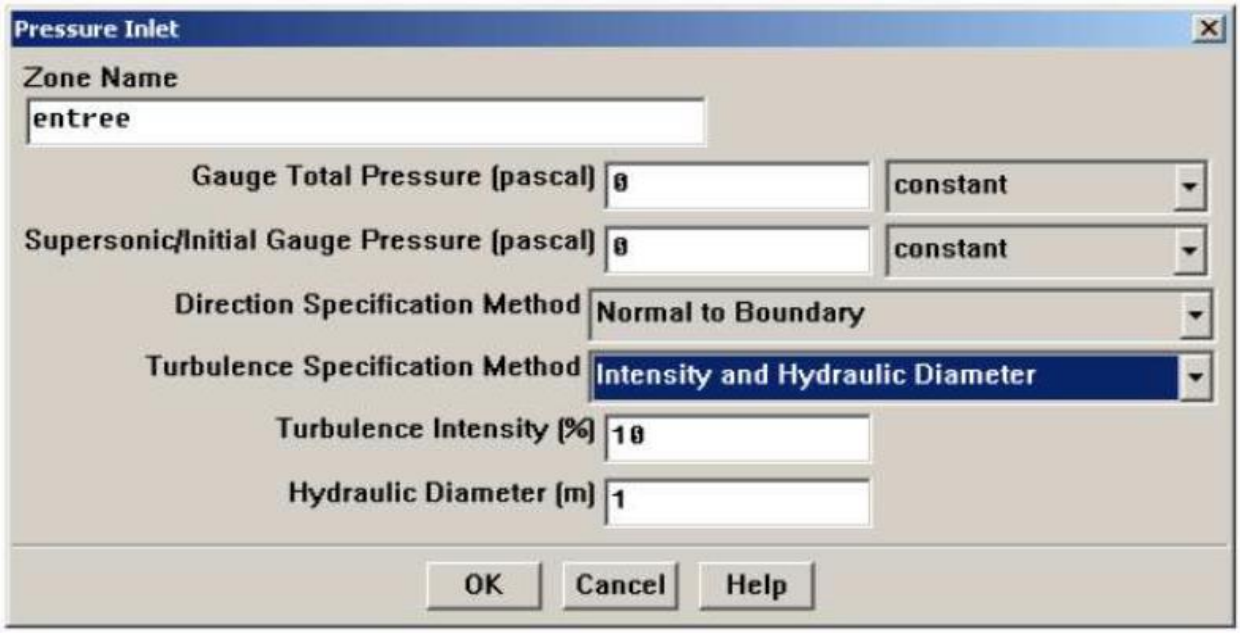


Figure II.19 : Pressure inlet.

- **Mass Flow Inlet**

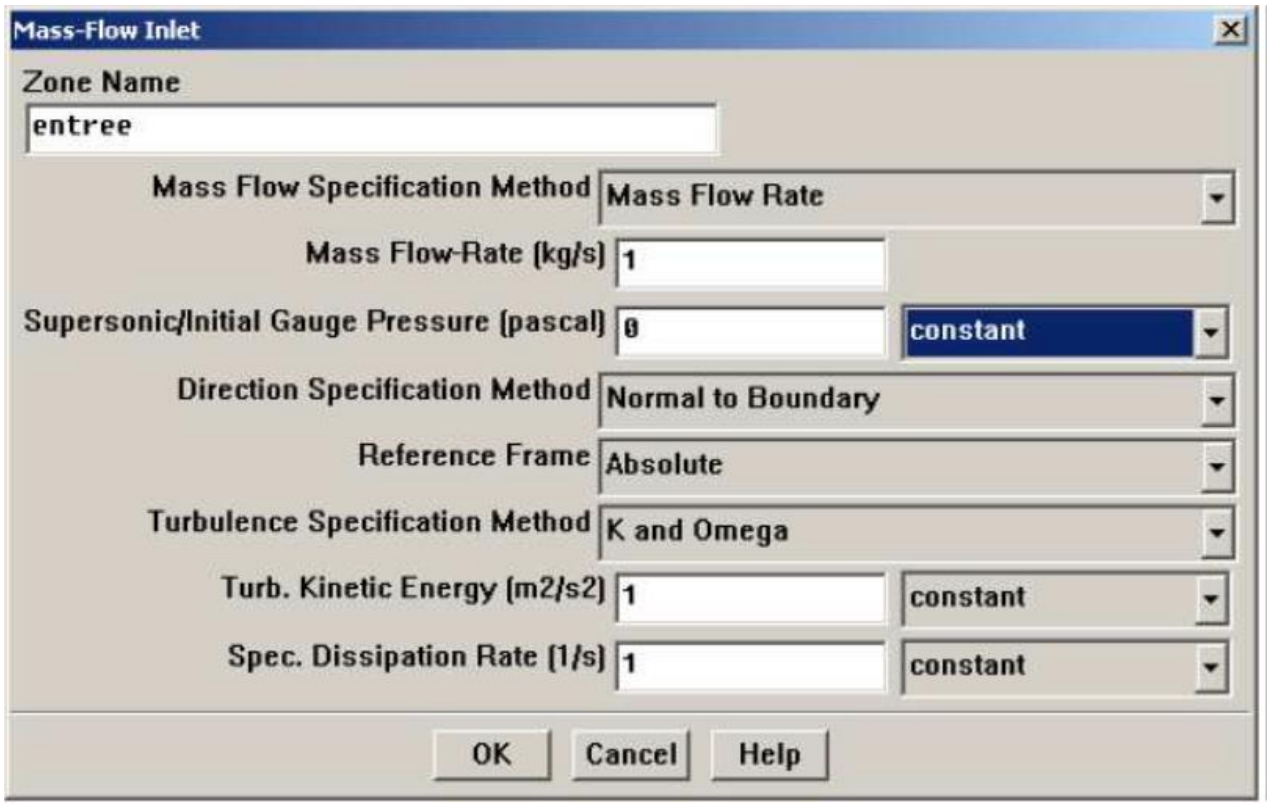
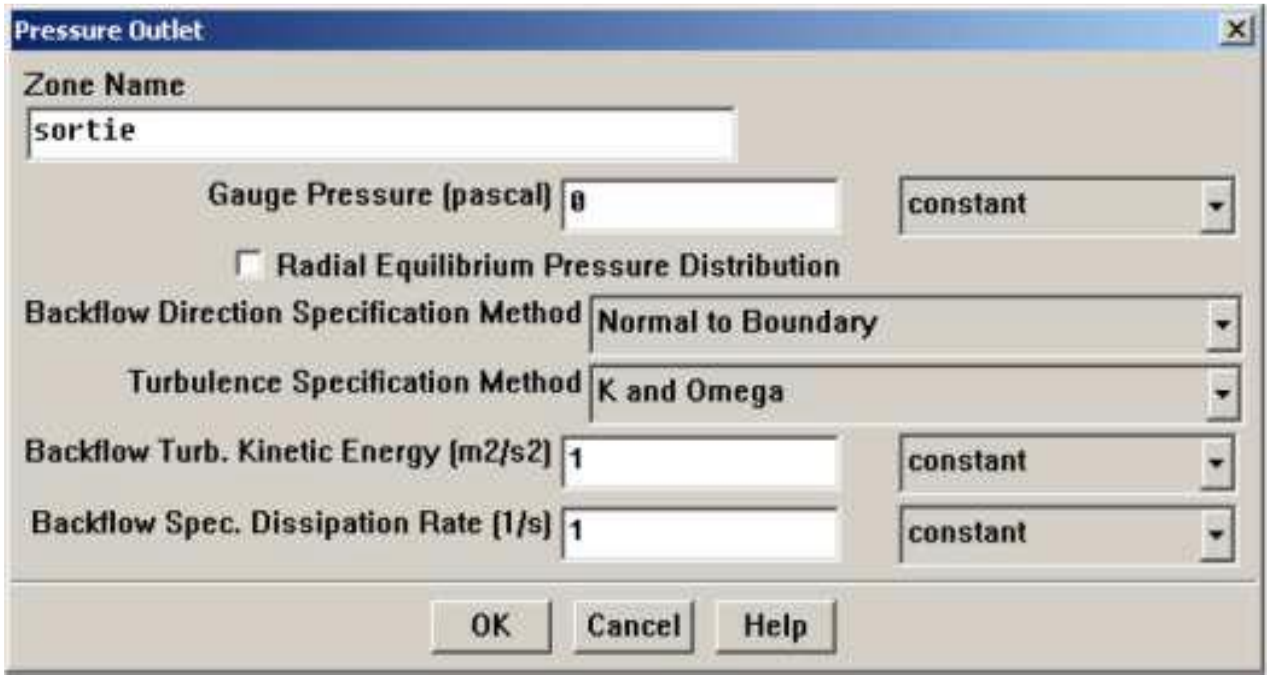


Figure II.20 : Mass Flow Inlet.

***** A known mass flow rate is required at the inlet, no need to use Mass Flow Inlet incompressible flow .

- **Pressure Outlet**

Specifies the output static pressure. The use of Pressure Outlet is used to set the output static pressure. Using the Pressure Outlets condition instead of Outflow often results in better convergence.



FigureII.21 : Pressure Outlet.

- **Outflow**

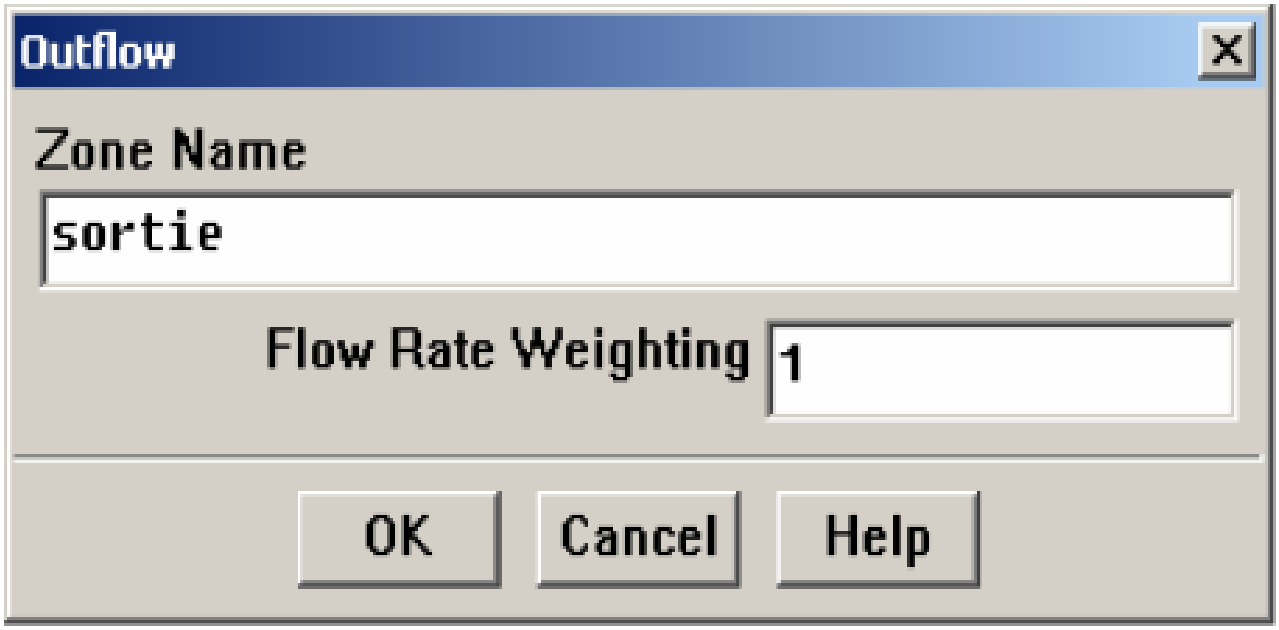


Figure II.22 :Out flow.

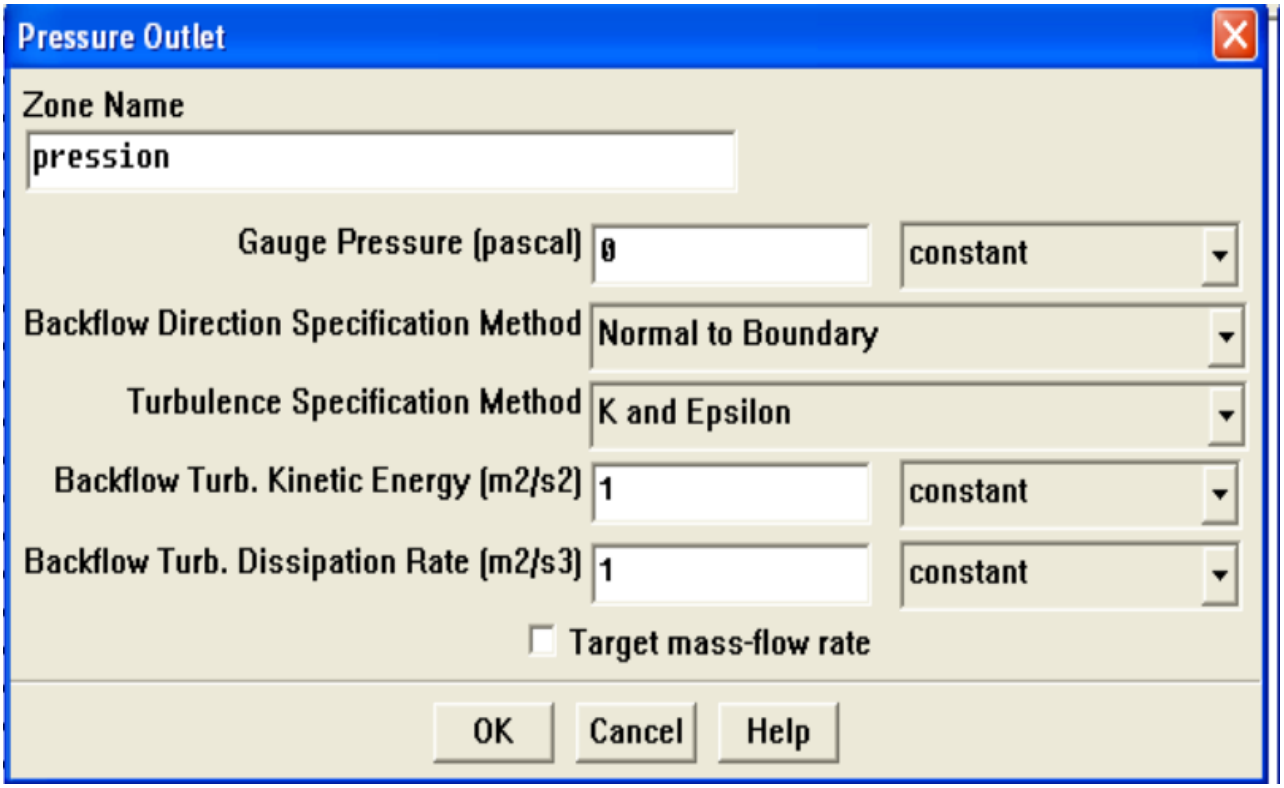


Figure II.23: output pressure interface.

II. 5 The finite volume method (FVM)

II. 5. 1 Introduction

Provide a general method for calculating the size and defining the surface of the three-dimensional objects that is used to calculate the size, including abstract mathematical forms that can be represented by equations.

This approach includes integrated, multifaceted knowledge, such as the quantification of forms with complex forms such as hearts, and involves the use of equations and insight to arrive at solutions such as the Gausian.

This method can be catalytic in understanding the concepts of differential and integral calculation of multiple variables. In addition, the quantification of solid forms is important for a variety of applications, such as photography, computational engineering, three-dimensional representation, and industrial industrialization.

This area has been explored for a long time, and there are multiple methods for calculating the size of three-dimensional forms, including three-dimensional integration, division, and the use of marginal forms.

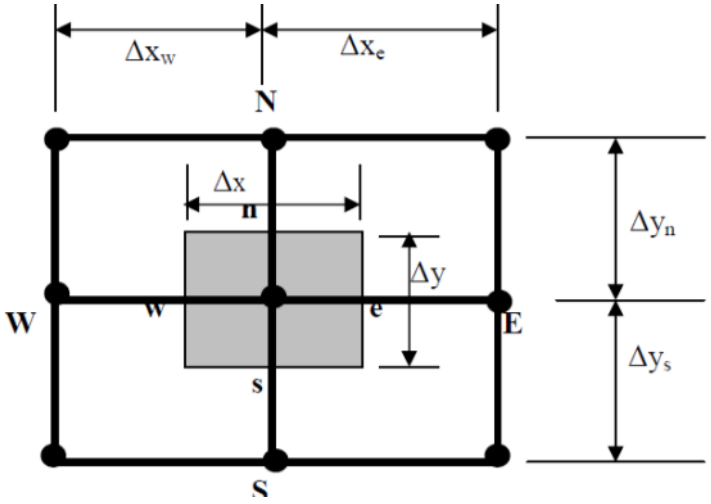
II. 5. 2 Definition of (FVM)

The finite volume method (MVF) is a discretization technique widely used, in the field of fluid mechanics where it emerged about twenty years ago. Since then, this method has undergone considerable development, not only for fluid mechanical modeling, but also for other fields of scientific engineering such as electromagnetism, thermal engineering, etc. In-depth mathematical analysis of the finite volume method has recently clarified the fundamental principles that make it

an effective discretionary technique. The main idea behind the formulation of finite volumes is intuitive and allowing the appearance of a clear physical interpretation of the phenomena studied.

This approach involves dividing the study area (Ω) into several elementary volumes so that each volume encompasses a main node (P). The differential equations of the model are then integrated into each control volume using an appropriate approximation scheme. The preservation principle is verified for both a finite control volume and an infinitesimal control volume.

The intellectual field is divided into several concrete volumes, where all the control volume is reduced, in a binary case, by four lines represented by small prescriptions indicating its frame (n to the north, e to the east, w to the west, s to the south) with respect to the "P" central box. High-altitude vehicles located on the north, east, west and south of the P vehicle are called N, E, W, and S. Intermediate dimensions such as pressure and rate are stored in the middle of specified volumes, while the velocity components are located on the limited sides of the volumes. All the size of the control is marked by the pieces x and y.As shown in figure: [8]



FigureII.24: Discretion in finite volumes of the area of study in the two-dimensional case.

Each finite section is crossed by a differential equation, the integral of which is calculated within that section. To this integral in the section, we use an approximation function to represent the unknown, such as linear, parabolic, exponential, or power, between two consecutive points.

Then we discretize the integral form on the entire area of study. This discretization results in an algebraic equation for values at nodal points. [19]

To facilitate the presentation of discretion by the finite volume method, we take the example of the two-dimensional fish equation:[8]

$$\frac{\partial}{\partial x}\left(\varepsilon\frac{\partial \emptyset}{\partial x}\right)+\frac{\partial}{\partial y}\left(\varepsilon\frac{\partial \emptyset}{\partial y}\right)=-\rho \qquad \text{II.9}$$

This equation is projected on a projection function that is equal to the unit. Then it is integrated into the finite volume corresponding to the P node, which gives:

$$\int_w^e\int_s^n \frac{\partial}{\partial x}\left(\varepsilon\frac{\partial \emptyset}{\partial x}\right)dxdy+\int_w^e\int_s^n \frac{\partial}{\partial y}\left(\varepsilon\frac{\partial \emptyset}{\partial y}\right)dxdy=\int_w^e\int_s^n -\rho dxdy \qquad \text{II.10}$$

The integration of the left part of (II.10) on the finite volume limited by (e, w, s, n) gives:

$$\int_w^e \int_s^n \frac{\partial}{\partial x}\left(\varepsilon \frac{\partial \phi}{\partial x}\right) dxdy + \int_w^e \int_s^n \frac{\partial}{\partial y}\left(\varepsilon \frac{\partial \phi}{\partial y}\right) dxdy = \left[\left(\varepsilon \frac{\partial \phi}{\partial x}\right)_e - \left(\varepsilon \frac{\partial \phi}{\partial x}\right)_w\right] \Delta y + \left[\left(\varepsilon \frac{\partial \phi}{\partial y}\right)_n - \left(\varepsilon \frac{\partial \phi}{\partial y}\right)_s\right] \Delta x \quad \text{II.11}$$

Choose a profile that expresses the variation in ϕ potential between neighboring nodes. The final result should contain only the node values.

$$\left(\varepsilon \frac{\partial \phi}{\partial x}\right)_e = \varepsilon_e \left(\frac{\phi_E - \phi_P}{\Delta x_e}\right) \quad \text{II.12}$$

$$\left(\varepsilon \frac{\partial \phi}{\partial x}\right)_w = \varepsilon_w \left(\frac{\phi_P - \phi_W}{\Delta x_w}\right) \quad \text{II.13}$$

$$\left(\varepsilon \frac{\partial \phi}{\partial y}\right)_n = \varepsilon_n \left(\frac{\phi_N - \phi_P}{\Delta y_n}\right) \quad \text{II.14}$$

$$\left(\varepsilon \frac{\partial \phi}{\partial y}\right)_s = \varepsilon_s \left(\frac{\phi_P - \phi_S}{\Delta y_s}\right) \quad \text{II.15}$$

The integral of the right part of equality (II.10) gives:

$$\int_w^e \int_s^n \rho dxdy = \rho_p \Delta x \, \Delta y \quad \text{II.16}$$

Thus the final algebraic equation after the integrals is written in the form :

$$\phi_P = a_E \phi_E + a_W \phi_W + a_S \phi_S + a_N \phi_N + S \quad \text{II.17}$$

With :

$$a_E = \varepsilon_e \frac{\Delta y}{\Delta x_e} \qquad a_w = \varepsilon_w \frac{\Delta y}{\Delta x_w}$$

$$a_n = \varepsilon_n \frac{\Delta x}{\Delta y_n} \qquad a_s = \varepsilon_s \frac{\Delta x}{\Delta y_s}$$

$$a_P = a_E + a_W + a_N + a_S$$

$$S = \rho_p \Delta x \, \Delta y$$

The reformulation of the result reached represents an obligatory equation linking the unknown in the main contract (P) to the unknowns in the adjacent contract (W, E, S, N).

This forced equation includes a system of linear equations in the form of a matrix [A].[ϕ]=[S]. This system is solved using direct or repetitive methods, such as the method of eliminating a spy and the method of a Sidle spy, respectively. The requirements of the non-homogeneous Derishle type are imposed on communications, and the requirements of the Newman type are imposed on other borders. [8]

II. 6 The Turbulence Phenomenon

II. 6. 1 Introduction

The technique of enhancing heat transfer through corrugated walls is a highly significant feature in the design of compact heat exchangers, as evidenced by numerous empirical studies and numerical analyses reported in the literature.

The physical process of increasing heat transfer stems from the fact that the corrugated shape of the surface disrupts the boundary layer formed and allows for continuous replacement of the fluid near the solid wall. Therefore, the optimal solution is to minimize pressure loss while maximizing heat transfer. [21]

II. 6. 2 Definition of Turbulence Phenomenon

Turbulence describes the chaotic state of a fluid, whether liquid or gas, where velocity exhibits vortices at every point, with constant variations in size, location, and orientation. Turbulent flows are characterized by their disorderly appearance, unpredictability, and the presence of multiple spatial and temporal scales.

They occur when the kinetic energy driving the fluid is much more intense than the viscous forces resisting its motion. In contrast, laminar flow is orderly. Understanding turbulence, though ancient, is still predominantly approached statistically, akin to statistical physics.

To illustrate the dominance of inertial forces over viscous forces in a flow, the Reynolds number is used, which must exceed a certain threshold. In natural environments, the Richardson number is often preferred over the Reynolds number as it accounts for density variation in compressible fluids.

An important aspect of turbulent flows is the energy cascade: large vortices break down into smaller ones, thus transferring energy from larger to smaller scales. However, this process is limited by molecular dissipation, which restricts velocity fluctuations.

Additionally, there is a phenomenon of vortices returning to larger structures, known as backscatter, facilitating the transfer of small structures to one or more larger ones. [21]

II. 6. 3 General characteristics of turbulence

Turbulence can be characterized by the following points: [21]

- Large-amplitude temporal and spatial fluctuations of all physical quantities (velocity components, pressure, etc.).

- Vertical structures of vastly different characteristic sizes, nested within each other, and interacting with one another.

- Each physical quantity exhibits an energy spectrum (i.e., spectrum of the square of the fluctuation) that is continuous and tends toward zero at high wave numbers (i.e., at the smallest spatial scales).

- Persistence of turbulence. Once initiated, turbulent flow tends to sustain itself: it continues to generate vortices to replace those dissipated. This is particularly true for turbulent flows with walls and for shear layers; this persistence is not related to the instability mechanisms of T-S waves in laminar flow.

- Mixing: Mixing in turbulent flow is much more efficient than mixing in laminar flow (i.e., through molecular diffusion).

II. 6. 4 K-ε turbulence model :

You have the ability to determine the actual turbulent viscosity of a fluid system by using the Turbulent Viscosity model. The latter includes two types of models: [15]

- The standard k-epsilon model: This turbulence model is available in Creo Flow Analysis. It is based on the formulation of turbulent kinetic energy k as follows:

$$\overline{-\rho u_i` u_j`} = \mu_t \left[\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_j}\right] - \frac{2}{3}\delta_{ij}\rho K \qquad \text{II.18}$$

The deformation tensor :

$$\frac{1}{2}\left[\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_j}\right]$$

Turbulent kinetic energy : $K = \frac{\overline{u_j`^2}}{2}$

The turbulent viscosity characterized by a speed$\sqrt{K}$ and a distance $L = \frac{K^{\frac{3}{2}}}{\varepsilon}$ is given by :

$$\upsilon_t = C_\mu \sqrt{K}\frac{K^{\frac{3}{2}}}{\varepsilon} = C_\mu \frac{K^2}{\varepsilon} \qquad \text{II.19}$$

With : C_μ =0.09

ε: is the dissipation rate

K and ε are obtained from these two functions:

a)Turbulent kinetic energy transport equation k:

$$\frac{\partial}{\partial x_j}(\rho K U_j) = \frac{\partial}{\partial x_j}\left[\left(\mu \frac{\mu_t}{\sigma k}\right)\frac{\partial K}{\partial x_j}\right] + G_k - \rho\varepsilon \qquad \text{II.20}$$

b) Transport equation of the dissipation rate ε of turbulent kinetic energy:

$$\frac{\partial}{\partial x_j}(\rho \varepsilon U_j)\Big| = \frac{\partial}{\partial x_j}\left[\left(\mu \frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial K}{\partial x_j}\right] + C_{1\varepsilon}\frac{\varepsilon}{K}G_K - C_{2\varepsilon}\rho\frac{\varepsilon^2}{k} \qquad \text{II.21}$$

Turbulent Prandtl numbers relating to turbulent kinetic energy and dissipation rate are represented by σ_k and σ_ε . The empirical constants are $C_{1\varepsilon}$ and $C_{2\varepsilon}$. The formula for the production term G_k is given by:

$$G_K = \overline{-\rho u_i` u_j`}\frac{\partial \overline{U_j}}{\partial x_i} = \rho v_t \left[\frac{\partial \overline{U_J}}{\partial x_j} + \frac{\partial \overline{U_j}}{\partial x_i}\right]\frac{\partial \overline{U_i}}{\partial x_j} \qquad \text{II.22}$$

All constants are grouped in the following table:

Table II.1 : K-ε turbulence model coefficients.

$C_{1\varepsilon}$	$C_{2\varepsilon}$	C_{μ}	σ_k	σ_{ε}
1,44	1.92	0.09	1	1.3

II. 7 The Trapezius Method

II. 7. 1 Introduction

The trapeze rule method is a numerical technique used to estimate the integral of a function by replacing it with a trapeze. This method is widely used in numerical analysis to approximate the area under a curve. In this paper, we will explore in detail this method and its applications in various mathematical and scientific fields.

II. 7. 2 Definition

The trapezium rule method consists in dividing the integration interval into several equal sub-intervals, then approxidizing the curve by trapezius formed by the points of these intervals. By making the sum of the areas of these trapezius, we obtain an estimate of the integral of the function over this interval.

This method allows a simple but effective approach for the numerical calculation of integrals.

II. 7. 3 Objective of the Method:

The main objective of the trapezoidal rule method is to provide a reasonable estimate of the integral of a function, especially when the function is difficult to integrate analytically.

This method also makes it possible to divide a complex integration problem into simpler steps, thus facilitating the numerical calculation process. In addition, it offers an intuitive approach to visualize the area under a curve geometrically.

Formally, if f(x) is the function to be integrated over the interval $[a, b]$and we divide this interval into nn subintervals of width h ($h = \frac{b-a}{n}$), then the estimation of the integral by the trapezoidal rule is given by:

$$\int_a^b f(x)dx = \frac{h}{2}[2f(a) + 2f(a+h) + 2f(a+2h) + \cdots + 2f(a+(n-1)h) + f(b)]$$

II.23

This formula is derived by approximating each subinterval with a trapezoid, hence the name of the method. The greater the number n of subintervals, the more accurate the approximation, as the trapezoids get closer to the curve of the function being integrated. [20]

II. 8 Conclusion

In conclusion, Fluent is computational fluid dynamics program and its Gambit pre-processor revolutionized the field of fluid dynamics simulation. By using the finite volume method, complex

engineering problems involving fluid flow and heat transfer can be analyzed and resolved accurately.

The semi-perverse phenomenon, characterized by distortion of network cells near corners or sharp edges, can insert errors in simulations if not properly calculated. In addition, the turbulent flow phenomenon, which occurs in high Reynolds flows, poses challenges in accurately predicting fluid behavior.

By understanding and integrating these phenomena into our simulations, we can improve the accuracy and reliability of computational fluid dynamics analyses, as we will use the k-ε turbulence model on which fluent is based in.

Chapter III :

Results and Interpretations

III. 1 Introduction

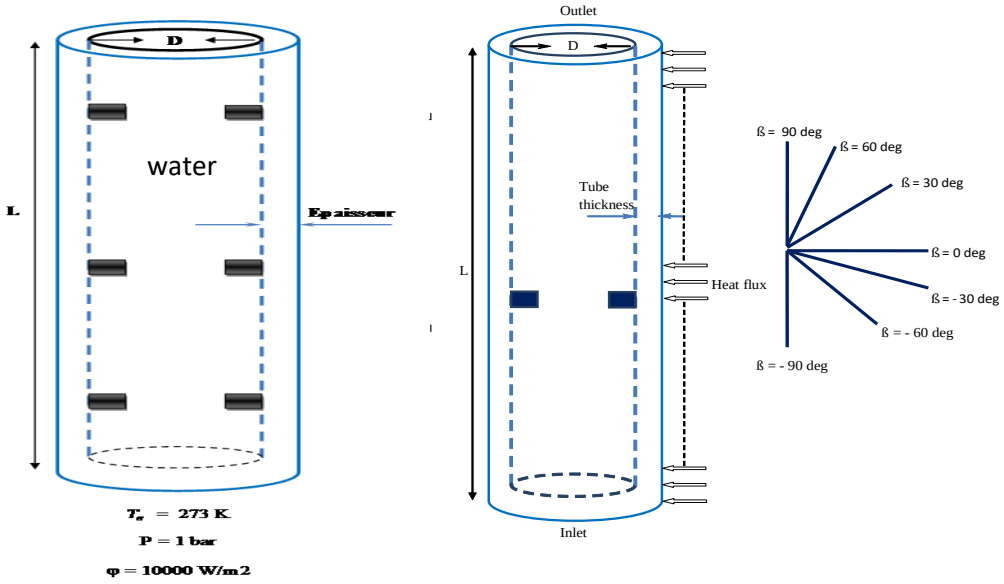
In this chapter, we present and discuss the results of the convection simulation in our cylindrical geometry, both vertically and horizontally oriented conduits, with and without a control element in three different positions, and conduits with a control element in only one position at x = 200 mm. We adhere to a logical arrangement in this study, beginning with presenting the results related to smooth conduits (without fins) oriented vertically and horizontally.

Afterwards, we will analyze the horizontal and vertical conduits with a control element in three positions (x = 50 mm, x = 200 mm, and x = 400 mm), as well as horizontal and vertical conduits with a control element in only one position at x = 200 mm.

III. 2 Problem Statement

The objective of this work is to study flow control to enhance heat transfer in cylindrical tubes. The use of control elements has been chosen as a technique to increase heat transfer in both horizontal and vertical tubes.

The geometry under investigation is a cylindrical copper tube with a length of 500 mm, an inner diameter of 10 mm, and a thickness of 1 mm. Fins in the form of circular rings with a thickness and width of 1 mm are inserted along the inner surface of the tube at different positions. The working fluid is water, transitioning from laminar to turbulent flow with Reynolds numbers of 1500 and 15000 respectively. See Figure 1.



FigureIII.1: conduit with control element (x=50,200,400) and x=200 mm).

III. 3 Mesh Selection

Mesh selection is a crucial step in numerical simulation. It ensures the independence of the solution obtained from the mesh used and avoids numerical errors. For this purpose, five different meshes were tested; their details are mentioned in Table (III.1).

The calculations were performed for a smooth vertical conduit under laminar flow with a Reynolds number Re = 1500 and for an inlet temperature and pressure of Te = 293 K and Pe = 1 bar respectively, with a heat flux density at the wall of q = 10000 W/m2. The heat transfer coefficients calculated using the numerical procedure corresponding to each chosen mesh are also presented in (Table III.2).

It can be observed that, starting from the third mesh (M 3), the value of the convective heat transfer coefficient remains constant. Therefore, meshes (M 1) and (M2) are not suitable, as there is an unjustified variation in the heat transfer coefficient. Finally, mesh (M 3) was selected for this numerical study because it yields the same results as meshes (M 4) and (M 5) but with a smaller size (see Table III.1). For the chosen mesh (M 3) illustrated in Figure III.1, a total of 270,000 elements were generated in the fluid zone and 30,000 elements in the solid zone.

Table III.1: Details of the different meshes tested.

Meshes	**fluid**	**solid**
M1	30000	10000
M2	120000	20000
M3	270000	30000
M4	30000	10000
M5	120000	20000

Table III.2: Average Convective Heat Transfer Coefficient for the Different Meshes.

Meshes	**M1**	**M2**	**M3**	**M4**	**M5**
$\bar{h}$ [w/m²K]	944.9524	908.4986	913.217	913.216	913.215

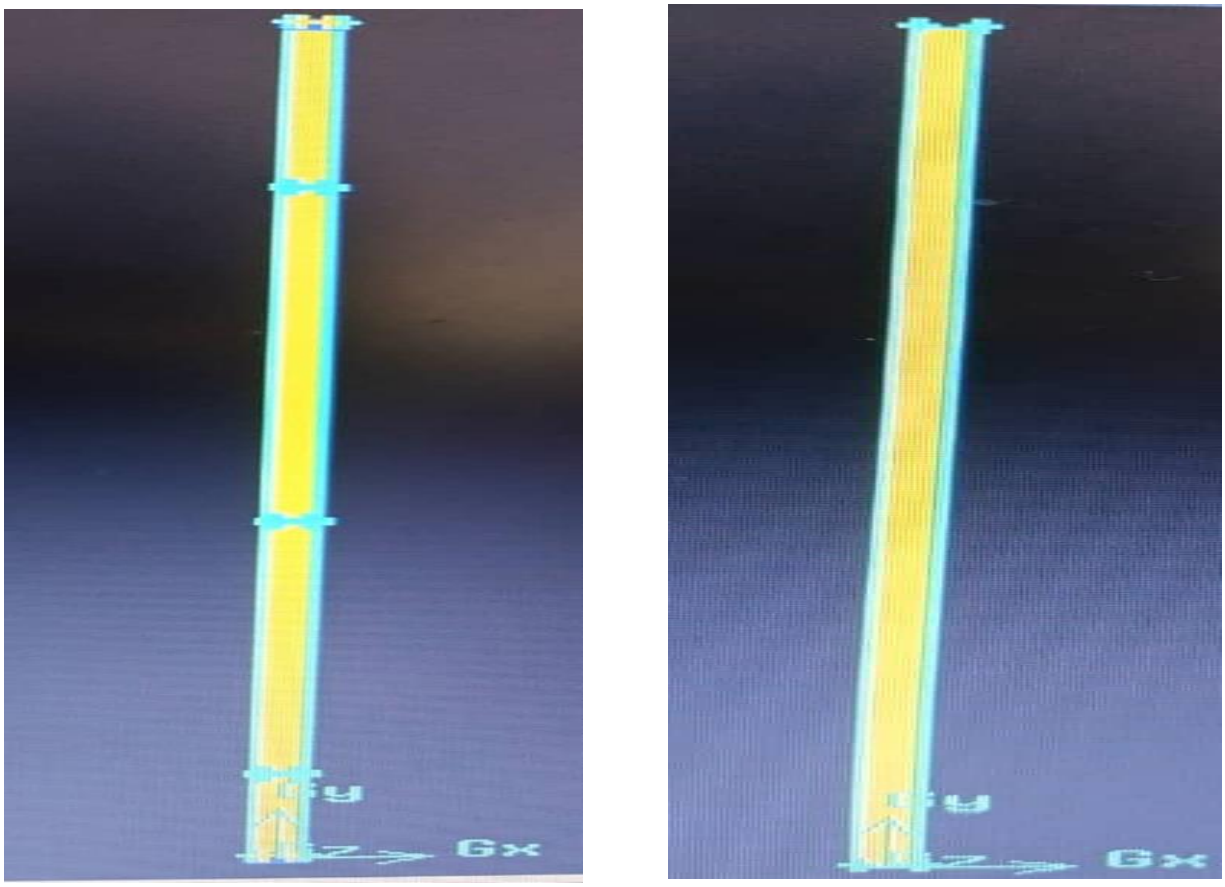


Figure III.2: Chosen Mesh.

Draw the mesh using Gambit

Validation of the numerical simulation procedure

The approach adopted to determine the convective heat transfer coefficient and the Nusselt number is as follows:

- The conduit is divided into several sections as shown in Figure (III.3).

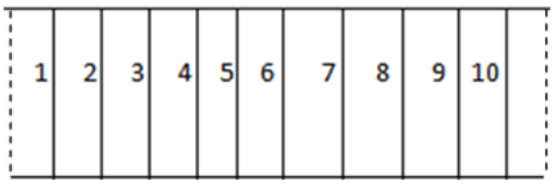


Figure III.3: Subdivision of the conduit section for the calculation of the local Nusselt number.

- In each section, we determine the heat flux density and the temperature at the wall.
- To calculate the reference temperature (average mixture temperature) in each section, we use the following equations:

$$T_m(x) = \frac{\int \rho u C_p T dA_c}{m C_p} \quad \text{III.1}$$

In each section, the local convective heat transfer coefficient is calculated using the following two equations:

$$h(x) = \frac{q_p(x)}{T_P(x) - T_m(x)} \quad \text{III.2}$$

$$\text{Nu(x)} = \frac{h(x).x}{\lambda} \quad \text{III.3}$$

- The average values of h and Nu are determined from the local values.

The Figure (4) shows a comparison of the locally calculated Nusselt number numerically in a smooth horizontal conduit under laminar flow with that obtained by the Shah and London correlation [23] for a Reynolds number Re = 1500. The Shah and London correlation for a thermally developed region and constant wall heat flux, as well as for laminar flow, is valid for:

$$\text{Nu}=1.953\left(Re.Pr.\frac{D}{L}\right)^{0.33} \quad \text{III.4}$$

Valid for: (Re.Pr.D/L) > 33.3

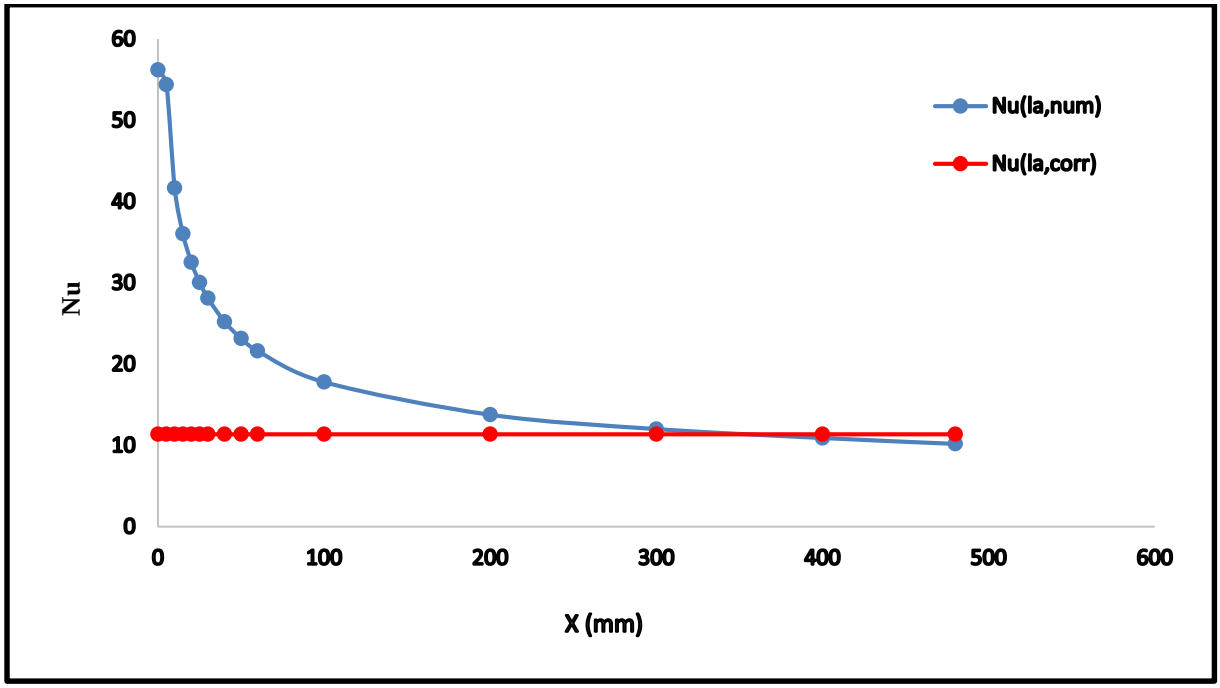


Figure III. 4: Local Nusselt numbers in a horizontal tube calculated by Fluent compared to the average Nusselt number determined by the correlation of Sahah and London [66].

It can be observed that from the distance x ≥ 200, the two curves converge and coincide. This zone corresponds to the region where the flow is thermally established. At the exit of the tube, the average values of the Nusselt number are almost the same for both procedures.

III. 4 Flow control for thermal transfer enhancement

III. 4. 1 Smooth vertical and horizontal conduits

Before examining flow control through the technique of thermal transfer enhancement (using vortex generators), it is useful to analyze the results obtained for the reference case of a vertical and horizontal tube without control elements (fins).

The boundary conditions for the results represented below are for the flow of water in the studied tube with an inlet temperature and pressure of 293 K and 1 bar, respectively. A constant heat flux of $10kW/m^2$ is imposed on the outer surface of the tube.

Two flow regimes have been studied: a laminar regime with a Reynolds number at the inlet Re = 1500 corresponding to a velocity u = 0.1507 m/s, and a turbulent regime with a Reynolds number Re = 15000 corresponding to an inlet velocity u = 1.507 m/s.

Figure 5 shows the variation of the local heat transfer coefficient in a smooth horizontal and vertical tube for a laminar regime.

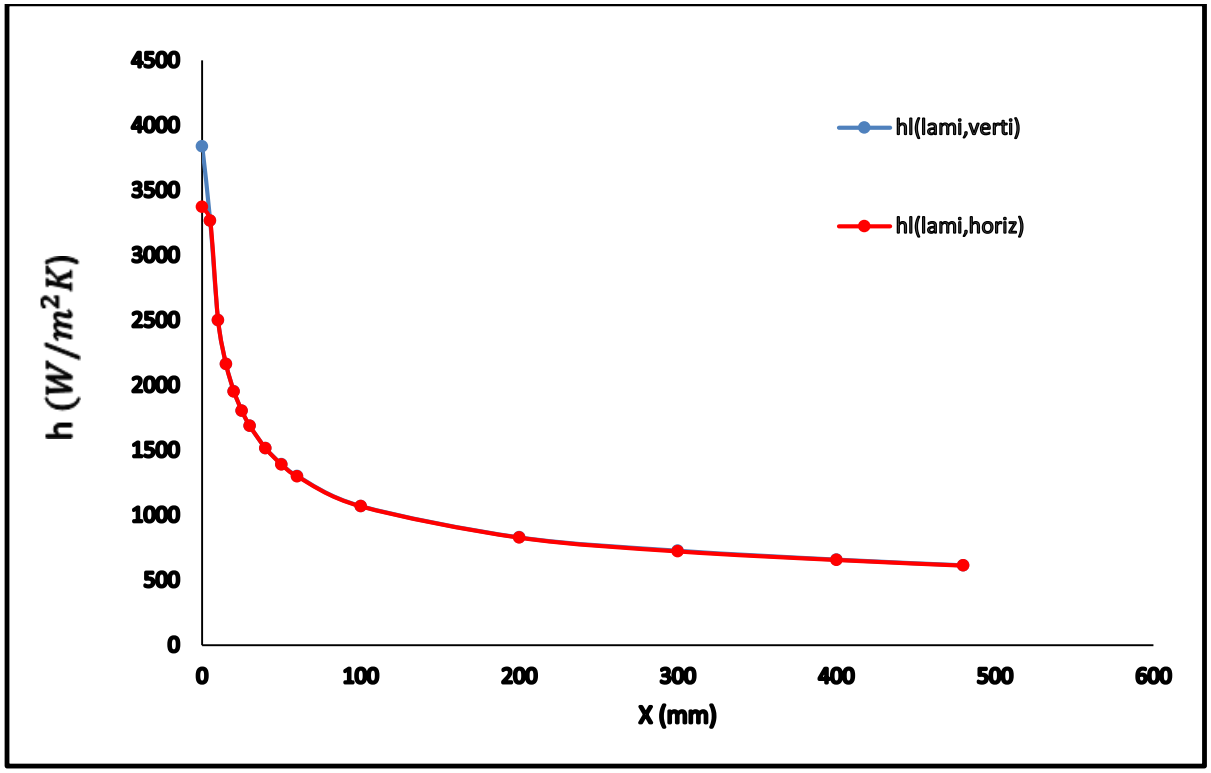


Figure III. 5: Local convective heat transfer coefficient in a smooth vertical and horizontal tube for a laminar regime.

We observe that the convective coefficient for laminar flow in a smooth horizontal tube is almost identical to the convective coefficient in a smooth vertical tube at almost all points .

Figure 6 shows the variation of the local heat transfer coefficient in a smooth vertical tube for a turbulent regime.

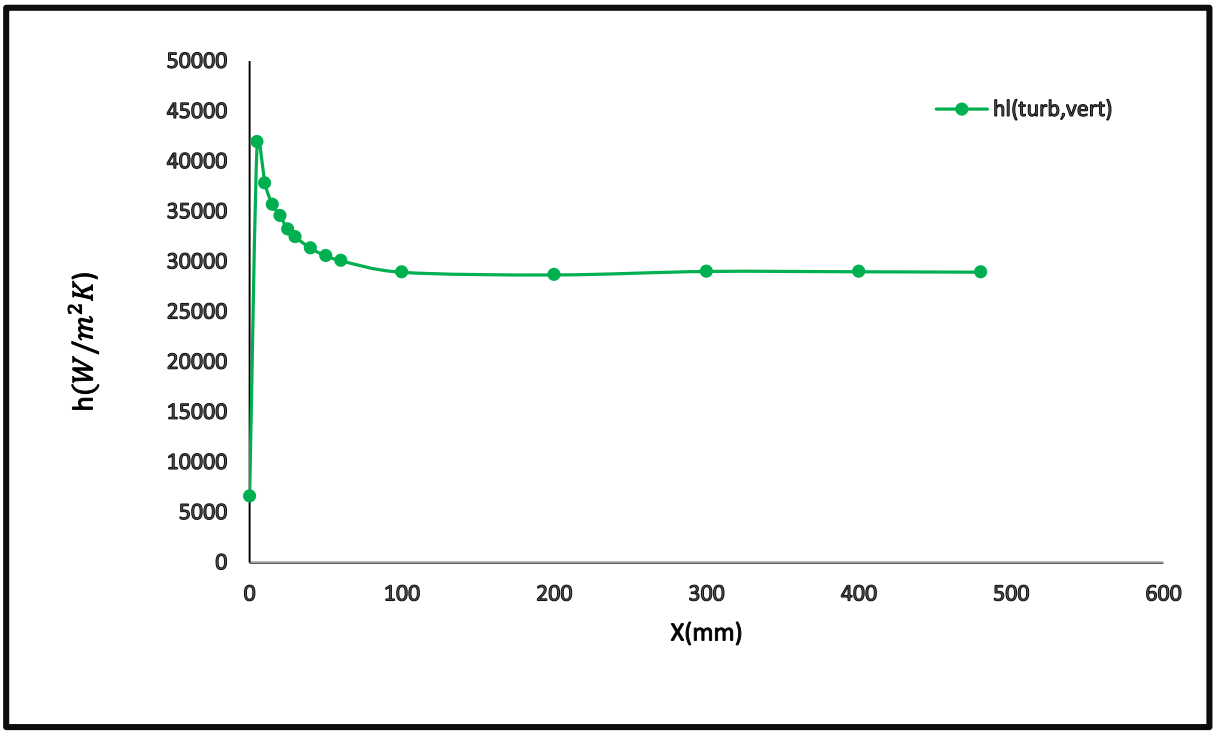


Figure III.6: Local convective heat transfer coefficient in a smooth vertical tube for a turbulent regime.

The figure 7 shows the variation of the local convective heat transfer coefficient in a smooth horizontal tube for a turbulent regime.

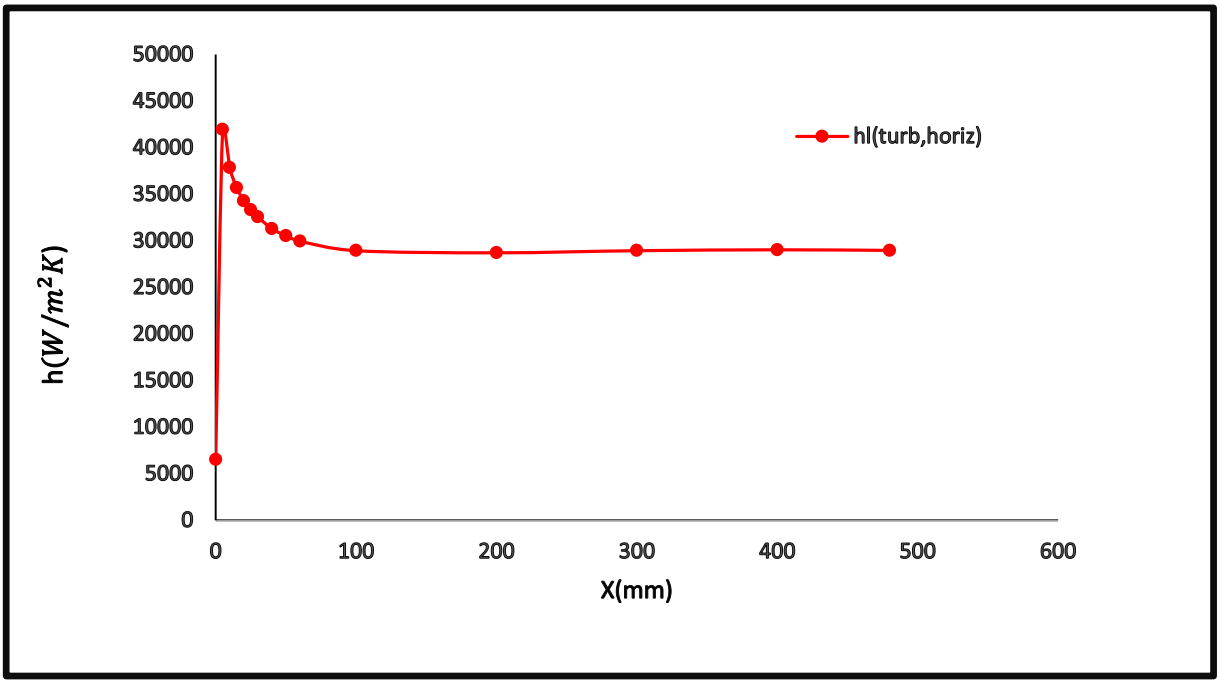


Figure III.7: Local convective heat transfer coefficient in a smooth horizontal tube for a turbulent regime.

We notice that the convective coefficient for the turbulent regime in a smooth horizontal tube is identical to the convective coefficient in a smooth vertical tube at all points.

The local convective coefficient for the turbulent regime is higher than for the laminar regime in both vertical and horizontal conduits at all points of x.

Figures 8 and 9 show the temperature and velocity fields in a vertical conduit for both laminar and turbulent regimes. Regarding the temperature field, the fluid heats up as it progresses along the tube. In the central part of the conduit, it is observed that for the laminar regime, the temperature does not undergo significant variations, unlike the turbulent regime. On the other hand, the cooling of the wall is better for the laminar regime. As for the velocity evolution, the slowing down of the flow at the wall leads to an increase in velocity in the central region to satisfy the principle of mass flow conservation.

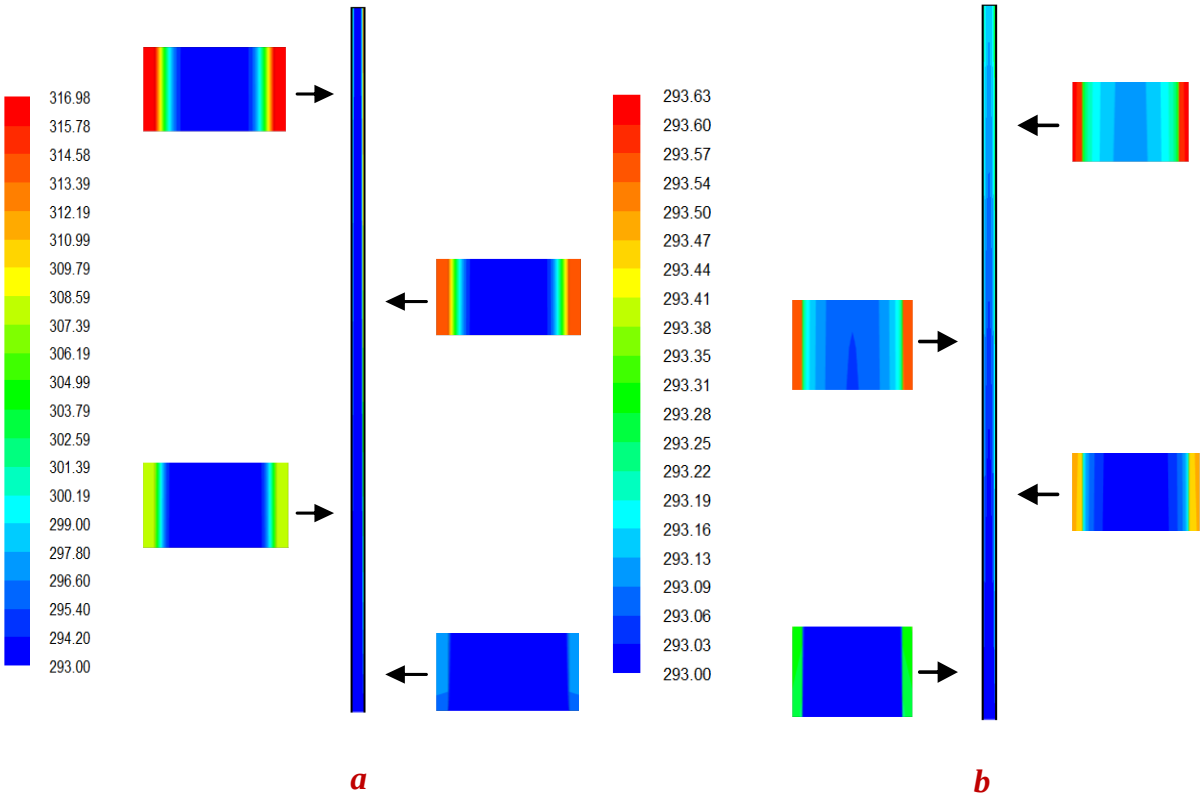


Figure III .8: Temperature fields for a smooth vertical conduit for laminar regime (a) and turbulent regime (b).

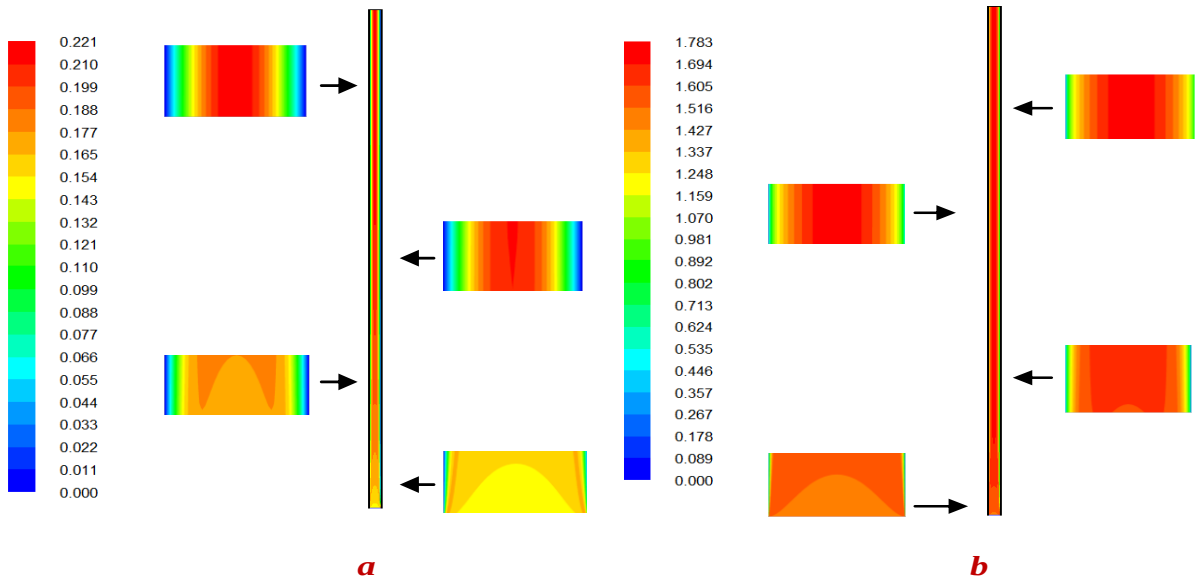


Figure III.9: Velocity fields for a smooth vertical conduit for laminar regime (a) and turbulent regime (b).

III. 4. 2 Study of control elements

III.4.2.1 Vertical and horizontal conduits with control element at x = 200 mm

Figure 10 shows the variation of the local heat transfer coefficient in a horizontal and vertical tube with a control element at x = 200 mm for a laminar regime.

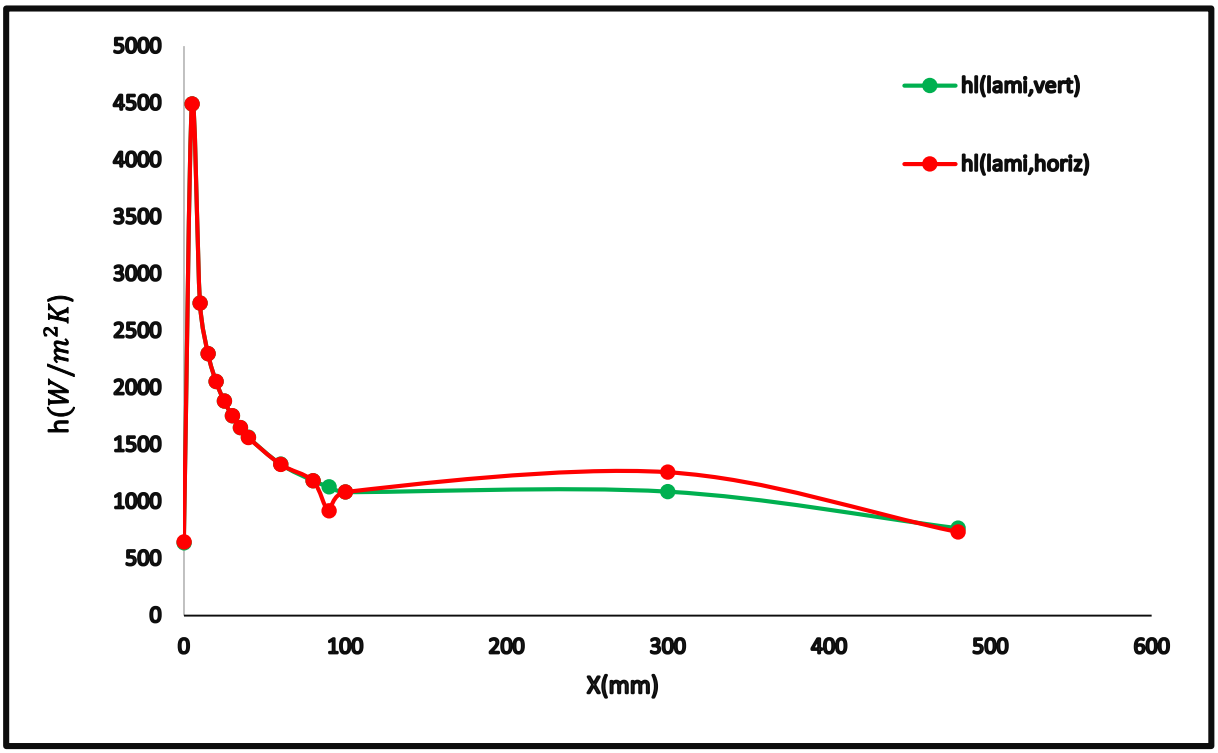


Figure III. 10: Local convective heat transfer coefficient in a vertical and horizontal tube with a control element at x = 200 mm for a laminar regime.

We observe that the convective coefficient for the laminar regime in a horizontal tube with a control element at x = 200 mm is almost identical to the convective coefficient in a vertical tube with a control element at x = 200 mm at all points except at x = 300 mm.

The figure 11 shows the variation of the local convective heat transfer coefficient in a horizontal and vertical tube with a control element at x = 200 mm for a turbulent regime.

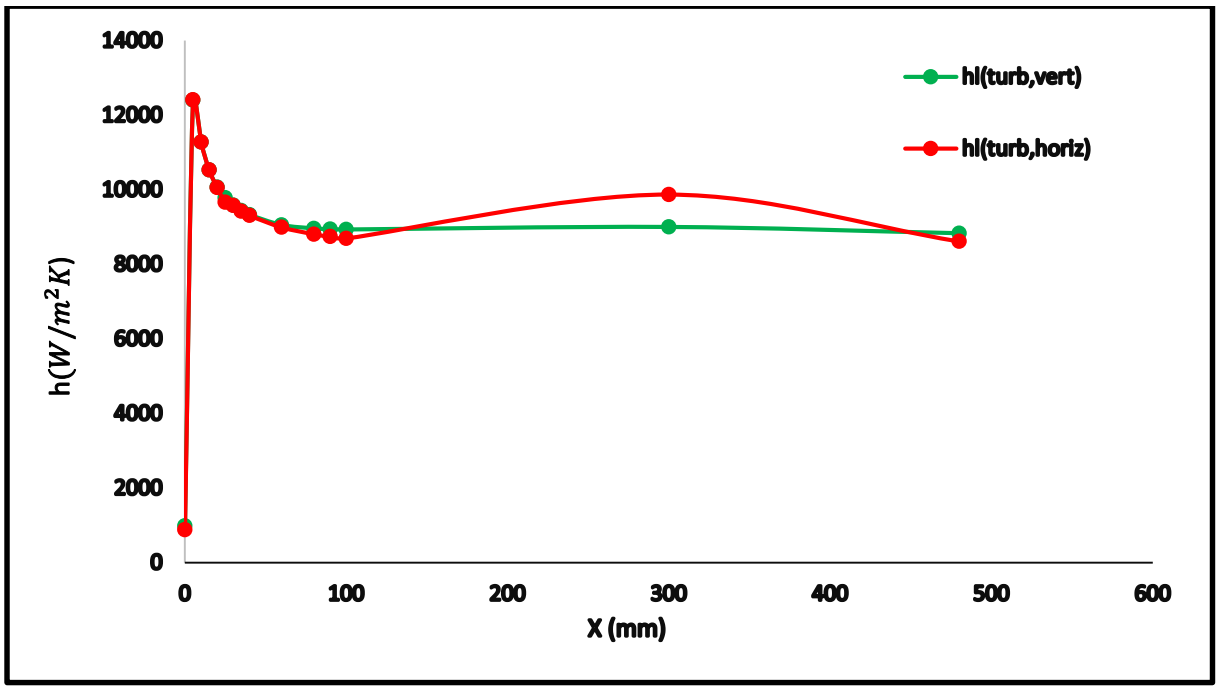
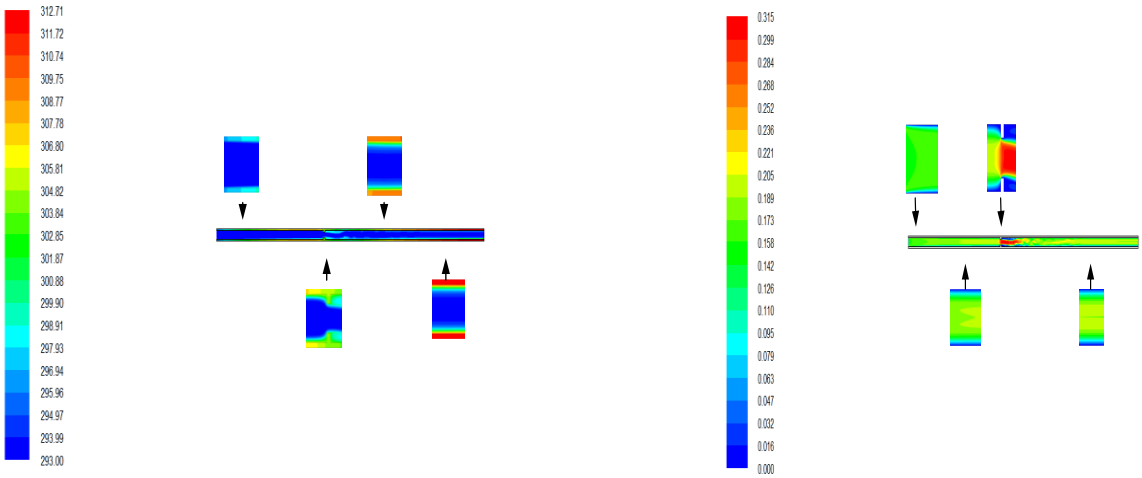


Figure III. 11: Local convective heat transfer coefficient in a vertical and horizontal tube with a control element at x = 200 mm for a turbulent regime.

We observe that the convective coefficient for the turbulent regime in a horizontal tube with a control element at x = 200 mm is identical to the convective coefficient in a vertical tube with a control element at x = 200 mm at all points except at x = 300 mm.

The local convective coefficient for the turbulent regime is higher than for the laminar regime in both vertical and horizontal conduits with a control element at (x = 200 mm) at all points of x.

Figure 12 shows the temperature and velocity fields for a horizontal conduit with a control element at x = 200 mm for a laminar regime.



FigureIII.12: Temperature and velocity fields for a horizontal conduit with a control element at x = 200 mm for a laminar regime.

The figure 13 shows a comparison of the variation of the local convective coefficient in a smooth horizontal tube and with a control element at x = 200 mm for a laminar regime.

We observe that the local convective coefficient in the horizontal conduit with a control element at x = 200 mm for the laminar regime is higher than the local convective coefficient in a smooth horizontal tube in the region from x = 100 mm to x = 500 mm.

The control element at x = 200 mm in the horizontal conduit improves heat transfer starting from the position x = 100 mm.

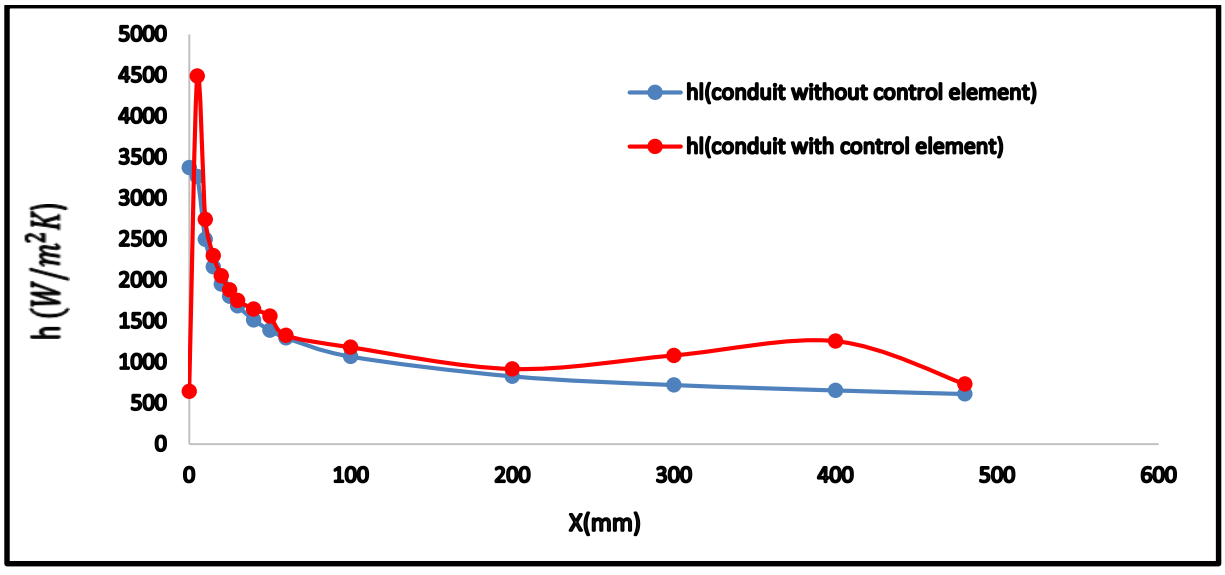


Figure III.13: Comparison of the variation of the local convective coefficient in a smooth horizontal tube and with a control element at x = 200 mm for a laminar regime.

III.4.2.2 Vertical and horizontal conduits with control elements(x=50, 200, and 400mm)

Figure 14 shows the variation of the local convective heat transfer coefficient in a horizontal and vertical tube with control elements at x = (50, 200, and 400 mm) for a laminar regime.

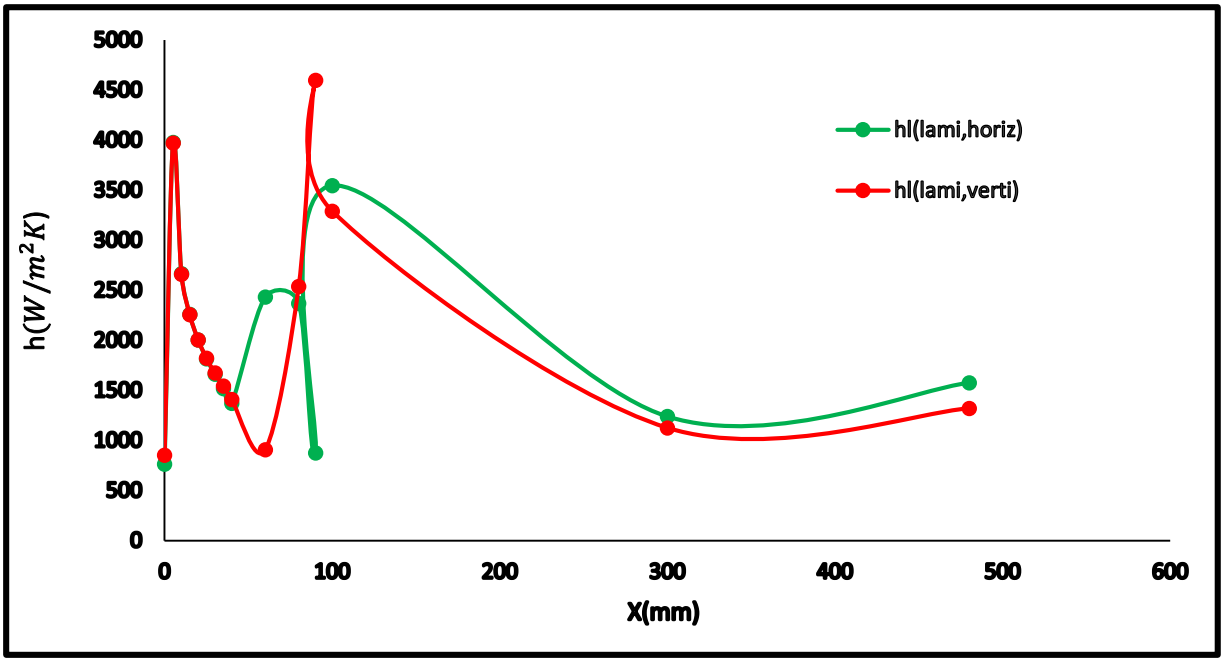


Figure III.14: Local convective heat transfer coefficient in a vertical and horizontal tube with control elements at (x = 50, 200, and 400 mm) for a laminar regime.

The local heat transfer coefficient for laminar flow in a horizontal pipe with control elements at (x = 50, 200, and 400 mm) is higher than the local heat transfer coefficient in a vertical pipe with control elements at (x = 50, 200, and 400 mm) at x = 60 mm.

The local heat transfer coefficient for laminar flow in a vertical pipe with control elements at (x = 50, 200, and 400 mm) is higher than the local heat transfer coefficient in a horizontal pipe with control elements at (x = 50, 200, and 400 mm) at x = 90 mm.

From x = 100 mm onwards, the local convection coefficient for laminar flow in a horizontal tube with control elements at (x = 50, 200, and 400 mm) is higher than the local convection coefficient for laminar flow in a vertical tube with control elements at (x = 50, 200, and 400 mm).

In the case of a pipe with a control element at x = 200 mm, the local convection coefficient is identical for both horizontal and vertical pipes. Therefore, to improve heat transfer using a control element at x = 200 mm, one chooses either the vertical or horizontal pipe. However, in the case with control elements at (x = 50, 200, and 400 mm), the local convection coefficient for the horizontal pipe is higher than that for the vertical pipe in the range from x = 100 mm to x = 500 mm. Hence, to improve heat transfer, it is preferable to use the horizontal pipe in this case.

Figure 15 shows the temperature field for laminar flow in a vertical tube with control elements at (x = 50, 200, and 400 mm).

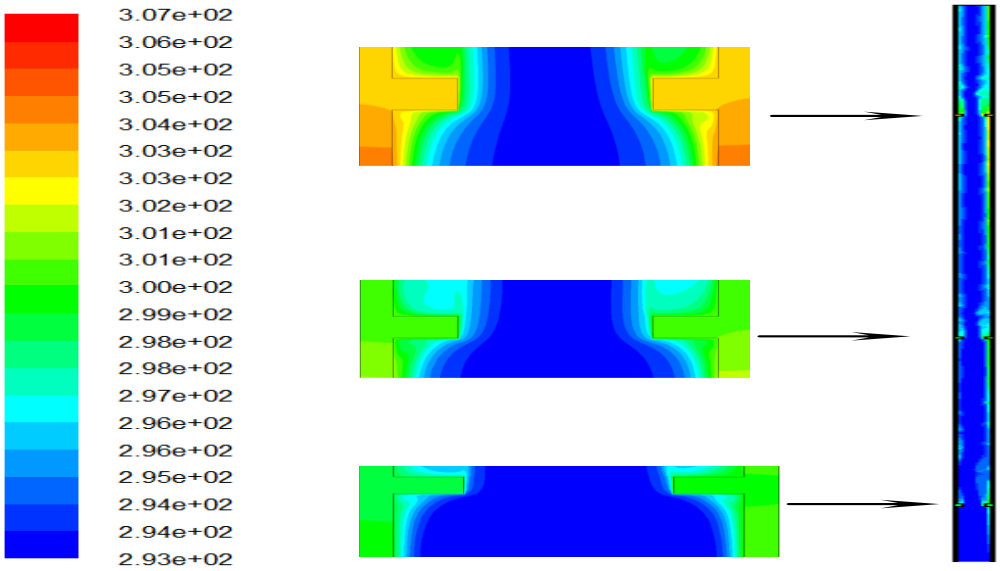


Figure III. 15: Temperature field for laminar flow in a vertical tube with control elements at (x = 50, 200, and 400 mm).

Figure 16 shows the variation of the local convective heat transfer coefficient in a vertical tube with control elements at (x = 50, 200, and 400 mm) for turbulent flow.

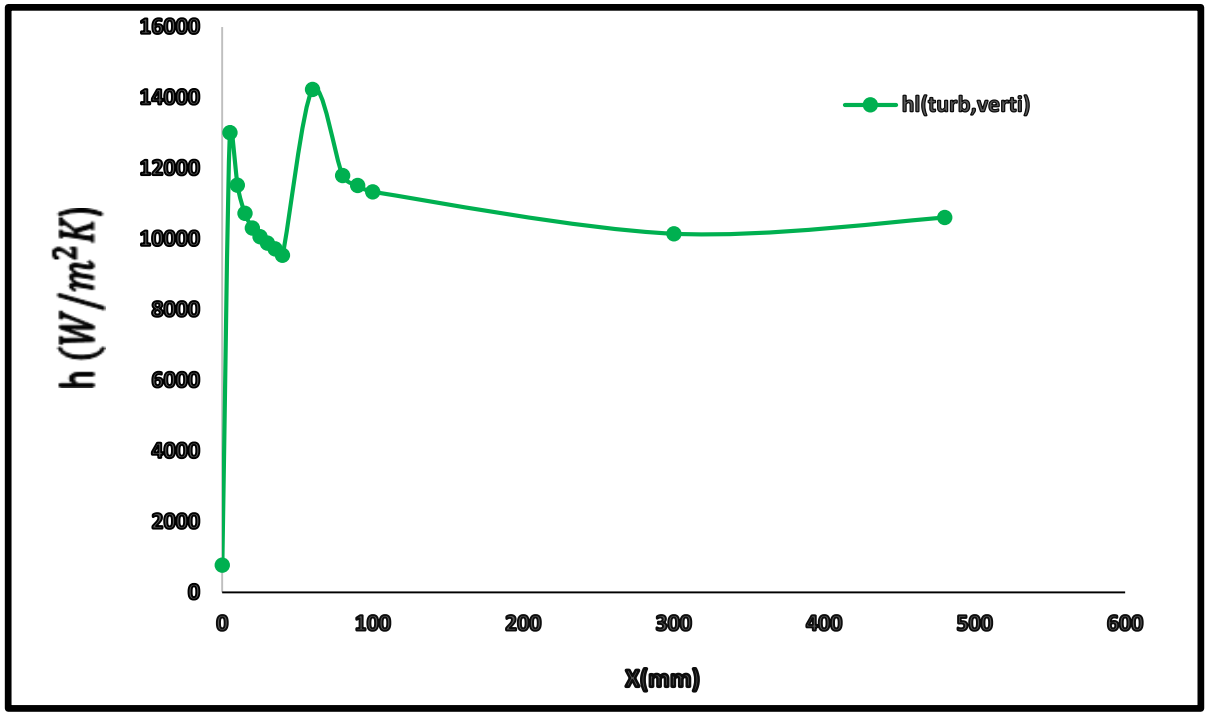


Figure III.16: Local convective heat transfer coefficient in a vertical tube with control elements at (x = 50, 200, and 400 mm) for turbulent flow.

Figure 17 illustrates the variation of the local heat transfer coefficient in a horizontal tube with control elements at x = (50, 200, and 400 mm) for turbulent flow.

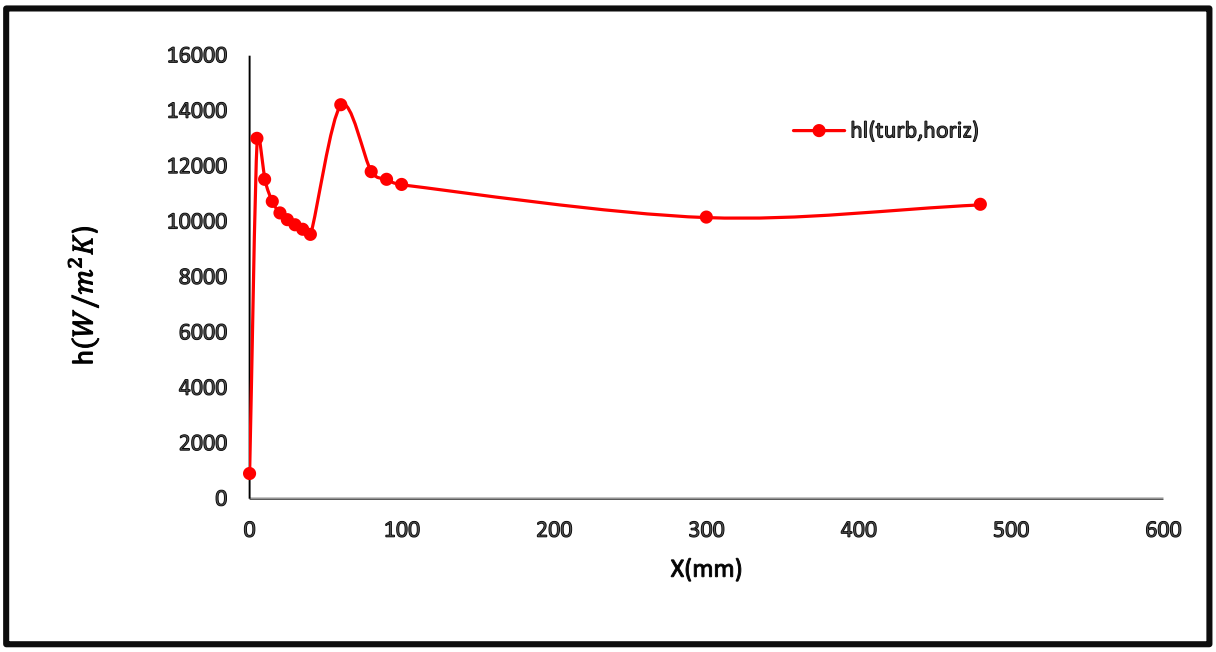


Figure III.17: Local convective heat transfer coefficient in a horizontal tube with control elements at (x = 50, 200, and 400 mm) for turbulent flow.

The convective heat transfer coefficient for turbulent flow in a horizontal tube with control elements at (x = 50, 200, and 400 mm) is identical to the convective heat transfer coefficient in a vertical tube with control elements at (x = 50, 200, and 400 mm) at all points.

The local convective heat transfer coefficient for turbulent flow is higher than for laminar flow in both the vertical and horizontal conduits with control elements at (x = 50, 200, and 400 mm) at all x points.

Figure 18 shows the variation of the local heat transfer coefficient in three tubes: the first tube is a smooth horizontal tube, the second tube is a horizontal tube with a control element at x = 200 mm, and the third tube is a horizontal tube with control elements at x = (50, 200, and 400 mm) for laminar flow.

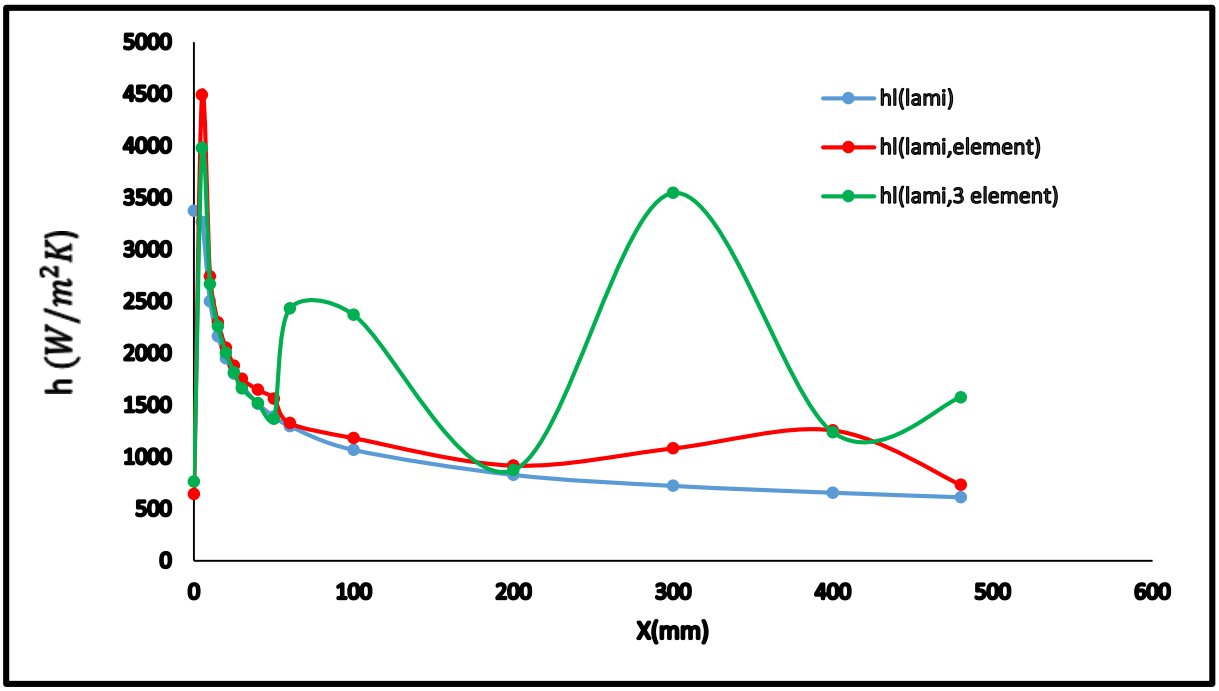


Figure III.18: Variation of the local heat transfer coefficient in three horizontal tubes for laminar flow.

The local convective heat transfer coefficient for laminar flow in the horizontal conduit with control elements (x = 50, 200, and 400 mm) is higher than the local convective heat transfer coefficient in a smooth conduit and a conduit with a control element at (x = 200 mm) starting from a distance of x = 40 mm.

The use of control elements in a horizontal tube enhances heat transfer for laminar flow whether using vortex generators at all three positions (x = 50, 200, and 400 mm) or using a single vortex generator at position (x = 200 mm). However, the best improvement for heat transfer in the horizontal conduit for laminar flow is achieved by using control elements at positions (x = 50, 200, and 400 mm).

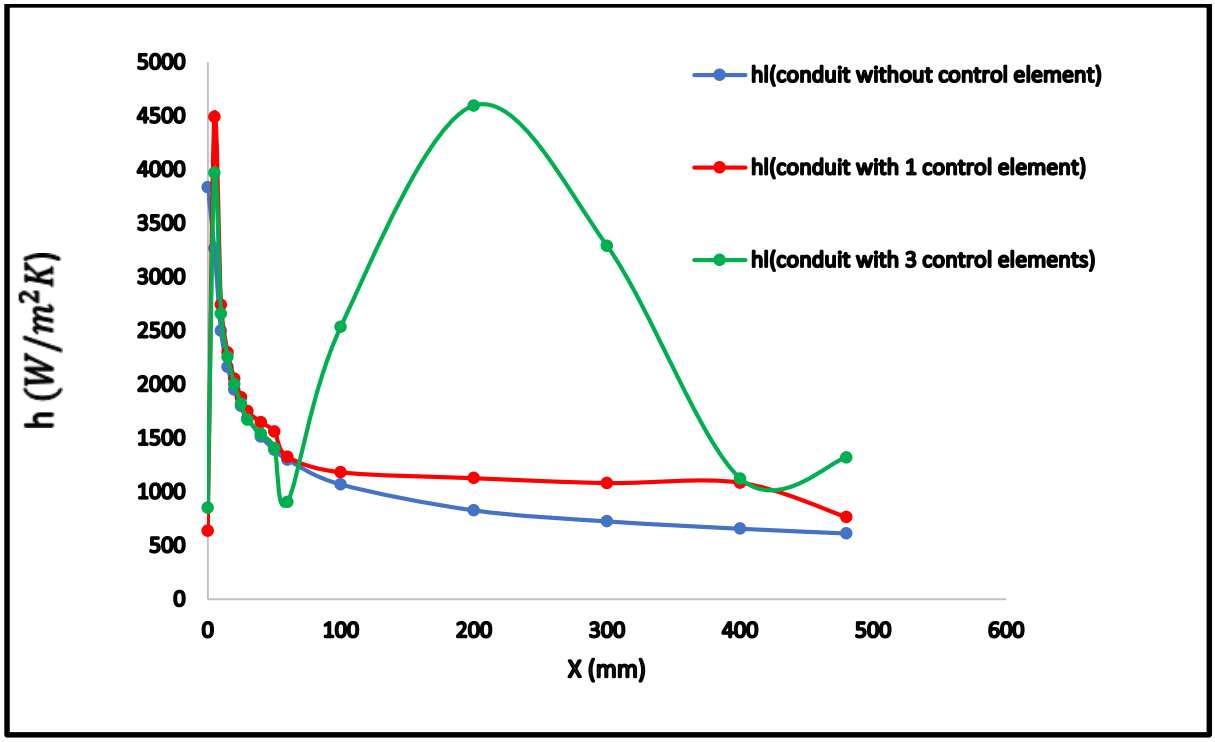


Figure III. 19: Variation of the local heat transfer coefficient in three vertical tubes for laminar flow.

The local convective heat transfer coefficient for laminar flow in vertical pipes decreases more than the convective local heat transfer coefficient in smooth pipes and pipes containing a control element at (x = 200 mm), starting from the distance x = 60 mm then rises again to peak at point (x=200 mm).

The use of control elements in a vertical tube enhances heat transfer for laminar flow, whether using vortex generators at all three positions (x = 50, 200, and 400 mm) or using a single vortex generator at position (x = 200 mm). However, the best improvement for heat transfer in the vertical conduit for laminar flow is achieved by using control elements at positions (x = 50, 200, and 400 mm).

Figure 20 shows the variation of the local heat transfer coefficient in three tubes: the first tube is a smooth vertical tube; the second tube is a vertical tube with a control element at x = 200 mm , the third tube is a vertical tube with control elements at x = (50, 200, and 400 mm) for turbulent flow.

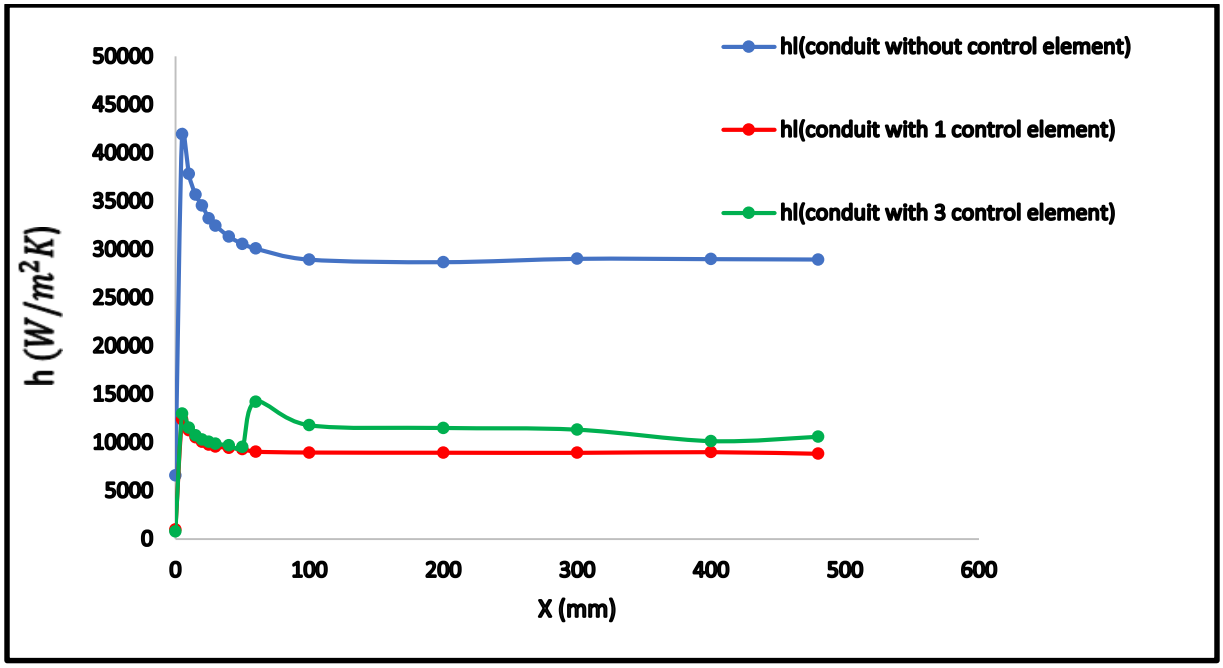


Figure III.20: Variation of the local heat transfer coefficient in three vertical tubes for turbulent flow.

The local convective heat transfer coefficient for turbulent flow in the vertical conduit with control elements (x = 50, 200, and 400 mm) and the local convective heat transfer coefficient in the conduit with a control element at (x = 200 mm) are lower than the local heat transfer coefficient in the smooth vertical conduit.

The use of control elements in a vertical tube does not improve heat transfer for turbulent flow, whether using vortex generators at all three positions (x = 50, 200, and 400 mm) or using a single vortex generator at position (x = 200 mm).

III. 5 Conclusion:

In this chapter, we conducted a comprehensive analysis of the impact of changes in the heat transfer coefficient in horizontal and vertical pipes under conditions with and without control elements. The study results showed that the heat transfer coefficient is significantly influenced by the distribution type (horizontal or vertical), as well as the presence of control elements. As for the Nusselt number in the laminar and turbulent flow regimes, it remains the same for both cases regarding the pipes.

In horizontal pipes, heat transfer was more stable and less affected by external environmental factors, whereas vertical pipes demonstrated an increase in heat transfer efficiency with the presence of control elements, which contributed to improved heat flow and distribution at (x=50, x=200, x=400) in the laminar flow regime. while it doesn't enhance it in the turbulent flow regime at these locations, resulting in lower heat transfer efficiency compared to the smooth pipe case.

Conversely, in the case of smooth pipes, whether in the laminar or turbulent flow regime, the heat transfer efficiency in both types of pipes is weak, with increasing differences in thermal performance between horizontal and vertical pipes.

In summary, this study underscores the importance of selecting the appropriate design and utilizing control elements to achieve optimal thermal performance. The findings highlight the necessity of considering operational and environmental conditions when designing thermal systems to ensure effective and efficient heat transfer.

General Conclusion

In conclusion, this study sheds light on the pivotal role of numerical simulation models in analyzing fluid flow dynamics and heat transfer phenomena within pipe systems. By exploring the behavior of viscous, incompressible fluids in both horizontal and vertical pipes, we underscored the significant impact of employing control elements to enhance heat transfer rates.

Utilizing the Fluent software, we conducted simulations that provided comprehensive insights into the factors influencing heat transfer enhancement. Our results clearly demonstrate that integrating control elements in both laminar and turbulent regimes leads to significant improvements in heat transfer rates across horizontal and vertical pipe systems.

Moreover, the positioning of these elements within the pipes exerts varying effects on heat transfer enhancement. These insights provide valuable guidance for optimizing the design of cooling and heating systems across diverse engineering applications, thereby enhancing the overall thermal efficiency of these systems.

This study highlights the necessity of integrating numerical simulation models as indispensable tools in fluid mechanics and heat transfer studies. Such integration can contribute to meaningful advancements in improving the performance and efficiency of thermal and hydraulic systems across various industrial domains.

BIBLIOGRAPHIC REFERENCES

[1] Yunus A, Ḉengel. Heat Transfer a Pratical Approach, 2002.

[2] Yunus A, Ḉengel. Fluid Machanics a fundamental and Application, 2006.

[3] M.Taki AL Kamil .Heat transfer a Basic Principles, 2019.

[4] J. E. Fowcs Williams, S. Rosenblat, J. T. Stuart, Transition from laminar to turbulent flow, Journal of Fluid Mechanics, 2006 ; 39 ; 433-442.

[5] Gordan J, Kumar R, Gupta A.Introduction to Acrospace flight Vehicles, 2024.

[6] Jannot Y, Heat Transfer, Cours Ecole des Miws Naacy, 2012.

[7] Sparrow, E. M., and Faghri, M. (August 1, 1980). "Fluid-to-Fluid Conjugate Heat Transfer for a Vertical Pipe-Internal Forced Convection and External Natural Convection." ASME. J. Heat Transfer. August 1980 ; 102(3) : 402–407.

[8] Merimech I, Etude numérique de la convection dans les tubes : contrôle des écoulements et optimisation des transferts de chaleur, Thèse de Dotorat, Université Batna, Algerie.

[9] Babar S S, Devade K D. Review on heat transfer enhacement with twisted tape, Mechanical Engineering (Indira College of Engineering and Management, Pure).

[10] Mathieu Legay. Intensification des processus de transfert de chaleur par ultrasons, vers un nouveau type d'échangeur de chaleur : l'échangeur vibrant, Thèse de Dotorat, Université de Grenoble, France.

[11] Poskas, Robert T, Boundary Layer Heat Transfer, Thermopedia of Thermodynamics Heat & Mass Transfer, and Fluids Engineering, 2011.

[12] Mehendale, S.S. (2018). Single-Phase Heat Exchangers. In : Handbook of Thermal Science and Engineering. Springer, Cham. https://doi.org/10.1007/978-3-319-26695-4_21.

[13] Jan H.K. Haertel a, Kurt Engelbrecht a, Boyan S. Lazarov b, Ole Sigmund b , Topology optimization of a pseudo 3D thermofluid heat sink model,International Journal of Heat and Mass transfer ,2018 ;121 ; 1073-1088.

[14] Doubbi Bounoua Karim, Analyse de la convection ws Chauffee Discretement par Le Bas Theme de Master, Université Mostaganem, Algerie.

[15] KORIB Kamel, Etude des écoulements autour des obstacles étude de simulation, Théme de Master, Université TLEMCEN, Algerie.

[16] Moulin N, emse .fr. Des equation de bilan a leur resoulution des element finis.

[17] HADJ,H, S ;Influence de la géométrie d'un orifice de soufflage sur la structure d'un JET LIBRE , Mémoire de Magister, Université de CHLEF.

[18] Gambit 2.2 Tutorial Guide, site : https://wp.kntu.ac.ir/mojra/CFD-gambit22tutorial.pdf https.

[19] Ferziger J H, Peri C M. Computational Methods for Fluid Dynamics, Springer-Verlag Berlin, Germany 2002.

[20] Guettaf Temam Halima Saadia, Integration Numérique et Applications, Universite BISKRA, Algerie.

[21] P. Chassaing, Turbulence en mécanique des fluides a Analyse de phénoméne en vue de sa modélisation à l'usage de l'ingenieur ,2000.

[22] Fluent Inc., (2006) Fluent 6.3 User's Guide, Fluent Inc., Lebanon, USA.

[23] Shah R K, London L. Laminar flow forced convection in ducts, Advances in Heat Transfer. London: Academic Press, 1978.

[24] Ferziger J H, Peric M. Computational Methods for Fluid Dynamics, Springer-Verlag Berlin, Germany 2002.