

FINAL YEAR PROJECT REPORT

ACADEMIC MASTER

Field: Mechanical Engineering

Specialty: Energy

Theme

Numerical simulation of a flow through a nozzle

Presented by:

- ❖ CHEHRI BAHAA EDDINE
- ❖ BOUGHAZI NADIR

Presented on 06/30/2026 before the panel consisting of :

President	Dr.GUERMAT Abdelkader	Université de Mostaganem UMAB
Examiner	Mr.SENOUCI Moussa	Université de Mostaganem UMAB
Supervisor	Dr. KHALDI Abdelaziz	Université de Mostaganem UMAB

Academic Year: 2025 / 2026

Acknowledgments

We would like to express my sincere gratitude to everyone who supported me throughout this research journey.

In particular, we extend our deepest appreciation to our supervisor, Dr. Abdelaziz Khaldi, for his guidance, kindness, and invaluable expertise, which were essential to the completion of this work.

We are also immensely grateful to all the professors in the Mechanical Engineering Department for their continuous encouragement and support. We hope this work reflects the high standard of knowledge and education they have instilled in us.

Abstract

This study investigates water flow through a nozzle installed in a pipe using numerical simulations performed with COMSOL Multiphysics. The objective was to evaluate the effects of three key parameters on flow behavior: Reynolds number, nozzle length, and expansion ratio.

Different Reynolds numbers (500–2500), nozzle lengths (0.042–0.21 m), and expansion ratios (3–7) were analyzed. The simulations provided detailed information about the velocity field, pressure field, streamlines, and recirculation zone within the flow domain.

The results showed that increasing the Reynolds number causes greater fluid acceleration and larger velocity and pressure gradients. A longer nozzle promotes a smoother and more stable flow development. In addition, increasing the expansion ratio (reducing the throat diameter) leads to higher maximum velocities and a greater pressure drop.

Overall, the study demonstrates that Reynolds number, nozzle length, and expansion ratio are critical factors governing the hydrodynamic behavior of flow through a nozzle and are important for the design and optimization of industrial nozzle systems.

Keywords: Numerical simulation, Nozzle, COMSOL Multiphysics, Reynolds number, Expansion ratio, Nozzle length, Velocity field, Pressure field.

المخلص

تتناول هذه الدراسة جريان الماء عبر فوهة (Nozzle) مثبتة داخل أنبوب باستخدام المحاكاة العددية المنجزة بواسطة برنامج **COMSOL Multiphysics**. وكان الهدف من الدراسة تقييم تأثير ثلاثة عوامل رئيسية على سلوك الجريان، وهي: عدد رينولدز، وطول الفوهة، ونسبة التمدد.

تم تحليل قيم مختلفة لعدد رينولدز (500–2500)، وأطوال الفوهة (0.042–0.21 م)، ونسب التمدد (3–7). وقد وفرت المحاكاة معلومات تفصيلية حول مجال السرعة، ومجال الضغط، وخطوط الانسياب، ومنطقة إعادة الدوران داخل مجال الجريان.

أظهرت النتائج أن زيادة عدد رينولدز تؤدي إلى زيادة تسارع المائع وارتفاع تدرجات السرعة والضغط. كما أن زيادة طول الفوهة تساعد على تطور الجريان بصورة أكثر سلاسة واستقرارًا. بالإضافة إلى ذلك، فإن زيادة نسبة التمدد (أي تقليل قطر الحلق) تؤدي إلى زيادة السرعة العظمى وحدوث انخفاض أكبر في الضغط.

بشكل عام، تُظهر هذه الدراسة أن عدد رينولدز، وطول الفوهة، ونسبة التمدد تُعد عوامل أساسية تتحكم في السلوك الهيدروديناميكي للجريان عبر الفوهة، ولها أهمية كبيرة في تصميم وتحسين أنظمة الفوهات المستخدمة في التطبيقات الصناعية.

الكلمات المفتاحية: المحاكاة العددية، الفوهة، **COMSOL Multiphysics**، عدد رينولدز، نسبة التمدد، طول الفوهة، مجال السرعة، مجال الضغط.

Résumé

Cette étude porte sur l'écoulement de l'eau à travers une tuyère (nozzle) installée dans une conduite, en utilisant des simulations numériques réalisées avec **COMSOL Multiphysics**. L'objectif était d'évaluer l'influence de trois paramètres principaux sur le comportement de l'écoulement : le nombre de Reynolds, la longueur de la tuyère et le rapport d'expansion.

Différentes valeurs du nombre de Reynolds (500–2500), de la longueur de la tuyère (0,042–0,21 m) et du rapport d'expansion (3–7) ont été analysées. Les simulations ont fourni des informations détaillées sur le champ de vitesse, le champ de pression, les lignes de courant et la zone de recirculation dans le domaine d'écoulement.

Les résultats ont montré que l'augmentation du nombre de Reynolds entraîne une accélération plus importante du fluide ainsi qu'une augmentation des gradients de vitesse et de pression. Une tuyère plus longue favorise un développement de l'écoulement plus progressif et plus stable. De plus, l'augmentation du rapport d'expansion (c'est-à-dire la réduction du diamètre du col) conduit à des vitesses maximales plus élevées et à une chute de pression plus importante.

Dans l'ensemble, cette étude montre que le nombre de Reynolds, la longueur de la tuyère et le rapport d'expansion sont des paramètres essentiels qui gouvernent le comportement hydrodynamique de l'écoulement à travers une tuyère. Ils jouent un rôle important dans la conception et l'optimisation des systèmes de tuyères utilisés dans les applications industrielles.

Mots-clés : Simulation numérique, Tuyère, COMSOL Multiphysics, Nombre de Reynolds, Rapport d'expansion, Longueur de la tuyère, Champ de vitesse, Champ de pression.

Table of Contents

General Introduction	1
I. CHAPTER I: Literature review	2
1.1 Introduction:	1
1.2 Some examples of applications involving flow through a nozzle:	1
1.3 Literature review:	2
1.4 Conclusion.....	5
II. CHAPTER II: Reminders and definitions	6
2.1 Introduction:	7
2.2 Types of fluids:	7
2.2.1 Perfect fluid:	7
2.2.2 Real fluid:.....	7
2.2.3 Compressible fluid:.....	7
2.2.4 Incompressible fluid:.....	7
2.3 Physical properties of fluids:	8
2.3.1 Density equation:.....	8
2.3.2 Viscosity:.....	8
2.3.3 Dynamic viscosity:	8
2.3.4 Kinematic viscosity:	9
2.3.5 Specific Gravity:	9
2.3.6 Debit:	9
2.4 Types of flows:.....	11
2.4.1 Permanent or stationary flows	11
2.4.2 Laminar or turbulent flow: Reynolds number	11
2.4.3 laminar flow.....	12
2.4.5 Turbulent regime:	13
2.5 Concept of pressure losses:	13
2.6 Linear pressure loss.....	13
2.7 Singular pressure loss.....	15
2.8 Types of Nozzles	16
2.8.1 Convergent Nozzles	16
2.8.2 Divergent Nozzles	17

2.8.3	Convergent-Divergent Nozzles.....	17
2.9	Conclusion.....	18
III.	CHAPTER 3 : Simulation, results, and interpretation.	19
3.1	Introduction	20
3.2	The model's geometry has certain conditions and limits:	20
3.3	Hypotheses	21
3.4	Equations of Motion (Incompressible Navier–Stokes Equations)	21
3.5	Simulation	22
3.5.1	Comsol Multiphysics:	22
3.5.2	Mesh Convergence Study:.....	22
3.5.3	First meshing method: Extremely fine triangular:	22
3.5.4	Second meshing method: extra fine triangle:.....	24
3.5.5	Third meshing method: Extremely fine quadrangle.....	26
3.6	Results and discussion	27
3.7	Study of the influence of the Reynolds number.....	27
3.8	Calculation of the Reynolds number.....	27
3.10	Study of the influence of the expansion report r:	37
3.11	Conclusion:.....	42
	General Conclusion.....	43
	References	44

LIST OF FIGURES :

Figure	Caption
Figure II.1	Fluid flow in the case of laminar flow.....12
Figure II.2	Fluid stream in the case of turbulent flow.....12
Figure II.3	Converging conical nozzle.....15
Figure II.4	Ajutage conique divergent.....16
Figure II.5	Convergent–Divergent nozzles.....17
Figure III.1	Representation of the problem geometry with boundary conditions.....19
Figure III.2	Representation of the mesh with an extremely fine triangular.....21
Figure III.3	Convergence of the calculation as a function of the number of iterations.....22
Figure III.4	Representation of the mesh with an extra fine triangle.....22
Figure III.5	Convergence of the calculation as a function of the number of iterations.....23
Figure III.6	Representation of the mesh with an extremely quadrangle mesh.....23
Figure III.7	Convergence of the calculation as a function of the number of iterations.....24
Figure III.8	Streamlines for $Re = 500$26
Figure III.9	Streamlines for $Re = 1000$26
Figure III.10	Streamlines for $Re = 1500$26
Figure III.11	Streamlines for $Re = 2000$27
Figure III.12	Streamlines for $Re = 2500$27
Figure III.13	Velocity field along the axis of symmetry of the pipe for different Reynolds numbers.....28
Figure III.14	pressure field along the axis of symmetry of the pipe for different Reynolds numbers.....29
Figure III.15	Streamlines for $Ln = 0.042$ m.....30
Figure III.16	Streamlines for $Ln = 0.084$ m.....30

Figure III.17	Streamlines for $L_n = 0.126$ m.....	31
Figure III.18	Streamlines for $L_n = 0.168$ m.....	31
Figure III.19	Streamlines for $L_n = 0.210$ m.....	31
Figure III.20	Velocity field along the axis of symmetry of the pipe for the influence of length.....	32
Figure III.21	pressure field along the axis of symmetry of the pipe for the influence of length.....	33
Figure III.22	Streamlines for expansion report $r = 3$	34
Figure III.23	Streamlines for expansion report $r = 4$	34
Figure III.24	Streamlines for expansion report $r = 5$	35
Figure III.25	Streamlines for expansion report $r = 6$	35
Figure III.26	Streamlines for expansion report $r = 7$	35
Figure III.27	Velocity field along the axis of symmetry of the pipe for different expansion ratios.....	36
Figure III.28	pressure field along the axis of symmetry of the pipe for different expansion ratios.....	37

Nomenclature

Latin Letters

Symbol	Description	Unit
D_1	Inlet diameter of the nozzle	m
D_2	Throat diameter of the nozzle	m
D	Diameter	m
L_n	Nozzle length	m
r	Expansion ratio (D_1/D_2)	—
Re	Reynolds number	—
U	Flow velocity	m/s
V	Velocity magnitude	m/s
P	Pressure	Pa
Q	Volumetric flow rate	m ³ /s
A	Cross-sectional area	m ²
x	Axial coordinate	m
y	Radial coordinate	m
ρ	Fluid density	kg/m ³
μ	Dynamic viscosity	Pa·s
ν	Kinematic viscosity	m ² /s
g	Gravitational acceleration	m/s ²

Greek Letters

Symbol	Description	Unit
α	Convergent/Divergent angle of the nozzle	°
μ	Dynamic viscosity	Pa·s
ν	Kinematic viscosity	m ² /s
ρ	Fluid density	kg/m ³
ΔP	Pressure difference (pressure drop)	Pa
ε	Convergence criterion (mesh convergence error)	—

General Introduction

Fluid flow through pipes with changing cross-sections is highly common in industrial applications. Among these devices, nozzles play a significant role due to their ability to accelerate the fluid and convert pressure energy into kinetic energy.

The primary objective is to examine the effects of key geometric and flow parameters, namely the Reynolds number, nozzle length, and expansion ratio, on the hydrodynamic behavior of the flow, including velocity and pressure distributions as well as the characteristics of the recirculation zone.

This thesis is organized into three main chapters. The first chapter provides a comprehensive literature review of previous studies related to flow through nozzles and highlights their various industrial applications.

The second chapter introduces the fundamental principles of fluid mechanics, presenting the main properties of fluids, flow regimes, and the different types of nozzles commonly used in engineering systems.

The third chapter focuses on the numerical modeling of the problem using COMSOL Multiphysics software. It describes the computational geometry, governing equations, boundary conditions, mesh generation, and the numerical procedure adopted. The obtained results are then presented and discussed in detail to evaluate the influence of the studied parameters on flow behavior.

Finally, the thesis concludes with a general conclusion summarizing the main findings of the study and emphasizing their relevance to the design and optimization of nozzle-based industrial systems.

I. CHAPTER I: Literature review

Chapter 1: Literature review

1.1 Introduction:

Flow nozzles are commonly used in various industries to measure the flow rate of liquids or gases passing through a pipeline. These devices rely on the principle of Bernoulli's equation to calculate the flow rate based on the pressure difference between the inlet and throat of the nozzle. Flow nozzles consist of a specially designed converging section that accelerates the fluid flow, followed by a throat section where the flow reaches its maximum velocity. By accurately measuring the pressure difference at these two points, flow nozzles can provide precise flow rate measurements.

Flow nozzles are preferred in many applications due to their simplicity, reliability, and accuracy. They are widely used in industries such as oil and gas, chemical processing, water treatment, and more. The design of flow nozzles allows for easy installation in existing pipelines, making them a versatile solution for flow measurement requirements. With their robust construction and minimal maintenance needs, flow nozzles are a cost-effective choice for many businesses.

1.2 Some examples of applications involving flow through a nozzle:

Flow through a nozzle shows up in a lot of engineering systems—anywhere fluid is accelerated by a change in cross-section. Here are some clear, practical examples across different fields:

One common application of flow nozzles is in the oil and gas industry, where they are used to monitor the flow of crude oil, natural gas, and refined products through pipelines. By accurately measuring flow rates, flow nozzles help operators maintain optimal flow conditions and prevent costly disruptions.

in a chemical plant, flow nozzles are used to measure the flow of chemicals and reagents to ensure efficient and high-quality production. Similarly, in water treatment facilities, they monitor the flow of water and wastewater to support effective treatment and proper resource management.

Flow nozzles are used in various industries such as HVAC, power generation, food and beverage, and aerospace. In all these fields, they ensure accurate flow measurement, helping maintain safe, efficient, and reliable operations.

Chapter 1: Literature review

1.3 Literature review:

J. Ohta and al (1993): The purpose of this study was to improve the performance characteristics of a convergent divergent nozzle for flashing expansion of initially subcooled hot water. The slip between the vapor and liquid along the divergent passage and the maximum nonequilibrium pressure drop at the nozzle throat cause a decline in the nozzle efficiency. To decrease the maximum pressure, drop, a nozzle with thin wires just upstream of the throat and a nozzle with a smaller convergent angle were manufactured and tested. Initially subcooled hot water at an inlet pressure of 0.47 MPa and inlet temperatures of 409.9 to 421.8 K (inlet subcooling's of 0.9–12.9 K) were used with the nozzle back pressures ranging from 5 to 101.3 kPa. The following results were attained: (1) the maximum nonequilibrium pressure drop at the throat decreased and the thrust coefficient increased by 10%; (2) the thrust coefficient was found to be independent of the inlet subcooling and exit stream-expansion ratio; (3) a correlation between the ratio of the exit pressure to the inlet pressure and the appropriate exit stream-expansion ratio was obtained empirically; and (4) the critical flow rate was found to be lower than that for an ordinary convergent-divergent nozzle. [1]

M. Shademan and al (2012): They investigated the effect of nozzle shape on turbulent flow using RANS simulations at a Reynolds number of approximately 50,000. Four nozzles—basic, extended, grooved, and annular—were analyzed using different turbulence models. The Reynolds stress (RSM) model yielded the most accurate results. The results indicate that the annular nozzle significantly increases turbulence and exit velocity but requires a higher inlet pressure. In contrast, the extended and grooved nozzles have a negligible effect on flow compared to the basic nozzle. [2]

[T. Luginsland](#) (2015): conducted a numerical study on the eddy effects of nozzle geometry (i.e., nozzle wall thickness d and nozzle length L) on the jet of a compressible swirl nozzle. For swirling swivel nozzles, the flow field changes significantly, mainly due to changes in the initial vortex number and enhanced centrifugal instability outside the nozzle wall. However, for a stationary nozzle, only small changes in the flow field were observed.[3]

Chapter 1: Literature review

LIAO Wen-tao¹ and DENG Xiao-yu² (2017): a mathematical model of the water jet flow field in a hydraulic cutting nozzle was developed. Through numerical simulation, the effects of the nozzle's convergence angle, outlet diameter, and cylindrical cross-section length on the water jet flow were determined, and the impact of these three factors on the water jet flow field was analyzed. The optimal nozzle parameters obtained through simulation were as follows: convergence angle 13°, cylindrical cross-section length 8 mm, and nozzle outlet diameter 2 mm. Under these optimal nozzle parameters, the hydraulic cutting achieves the best overall performance. [4]

Tao Zhang and al (2017): This study examines a pressure-swirl nozzle and how its design and operating conditions affect spray behavior. The orifice diameter (0.3–0.8 mm) strongly affects spray velocity and shape. A length-to-diameter ratio of 2 gives better, producing higher spray angles and velocities. Increasing injection pressure improves spray behavior only up to 1.5 MPa; beyond this, the spray angle stays constant. A relationship between Reynolds number and spray angle was established, showing that nozzle performance mainly depends on flow dynamics inside the orifice. Nozzle geometry (especially orifice size and L/D ratio) controls spray quality more than pressure at high operating conditions. [5]

M.H.H. Ishak and al (2019): The effect of nozzle spray shape towards the spray characteristic of hybrid biofuel blends is analyzed and compared with the standard diesel. Furthermore, verification and validation from both the experimental results and numerical results are also presented. The nozzle flow simulation results indicated that the fuel type did not affect the cavitation area vastly but were more dependent on the nozzle spray shape. In addition, the spray width of the elliptical nozzle shape was higher as compared to the circular spray. Moreover, as the backpressure increased, the spray width downstream increased as well. The spray tip penetration for the elliptical nozzle shape was shorter than the circular nozzle shape due to circular nozzles having circular nozzle shapes having smaller nozzle widths and less spray cone angles. Thus, this resulted in smaller aerodynamic drag. [6]

Tianwen Jiang and al (2022): They investigated improving the design of a straight conical nozzle by analyzing the effect of the convergence angle and nozzle neck length on flow performance. The

Chapter 1: Literature review

results show that a convergence angle of approximately 15° achieves optimal balance, with low frictional resistance and stable flow. Angles between 10° and 15° result in higher discharge efficiency and better jet quality. Additionally, a nozzle neck length three times the outlet diameter produces a longer and more stable jet. Overall, the optimal nozzle design features a convergence angle of 15° and a neck length three times the outlet diameter, ensuring better jet performance and a longer jet distance. [7]

Khan, A and Al (2024): By studying simulated sudden-expanding flows from closely spaced nozzles at acoustic Mach numbers, researchers gained valuable insights into velocity distributions and base pressure variations. Their study highlighted the importance of nozzle geometry and expansion ratios in shaping flow characteristics. Advances in computational fluid dynamics (CFD) have facilitated more comprehensive analyses. [8]

X. Zhu^{1,2†} and Al (2025): This study aims to improve the cutting efficiency of high-pressure abrasive water jets (AWJ) by optimizing both nozzle design and hydraulic parameters. CFD simulations showed that the length of conical nozzles has little effect on performance, while a 40° taper angle provides the best flow guidance. An SPH-based model was used to analyze the effects of pump pressure, flow rate, abrasive concentration, and nozzle diameter. Results revealed that there is an optimal abrasive concentration depending on operating conditions, and this optimal value increases with higher flow rates. Under constant pressure, increasing nozzle diameter raises flow rate, while cutting depth and volume increase then decrease, reaching maximum performance at a nozzle diameter of 4–5 mm. These findings offer a strong theoretical foundation for improving abrasive jet cutting technology [9].

Chapter 1: Literature review

1.4 Conclusion:

This chapter provided an overview of flow through nozzles and highlighted its importance in various engineering and industrial applications. The literature review demonstrated that nozzle geometry, operating conditions, and flow characteristics have a significant influence on performance, efficiency, and fluid behavior. Previous experimental, numerical, and computational studies showed that optimizing parameters such as convergence angle, throat dimensions, nozzle length, and operating pressure can greatly improve flow efficiency and jet quality. Therefore, the studies presented in this chapter establish a solid scientific foundation for further investigation and analysis of flow through nozzles, contributing to the development of more efficient and optimized nozzle designs.

II. CHAPTER II: Reminders and definitions

Chapter 2: Reminders and definition

2.1 Introduction:

Fluid mechanics is the study of fluids either in motion (fluid dynamics) or at rest (fluid statics). Both gases and liquids are classified as fluids, and the number of fluid engineering applications is enormous: breathing, blood flow, swimming, pumps, fans, turbines, airplanes, ships, rivers, windmills, pipes, missiles, icebergs, engines, filters, jets, and sprinklers, to name a few. When you think about it, almost everything on this planet either is fluid or moves within or near a fluid.[10]

2.2 Types of fluids:

Fluids are materials that can flow and adapt to the shape of any container they occupy. At equilibrium, they are unable to resist shear (tangential) or transverse forces.[11]

2.2.1 Perfect fluid:

A perfect liquid is an idealized concept characterized by zero viscosity and incompressibility, meaning it cannot be compressed. Such a fluid is purely theoretical and does not exist in real-world conditions.

2.2.2 Real fluid:

Unlike a perfect fluid, the motion of a real fluid involves energy losses in the form of heat due to viscous friction during its flow.

2.2.3 Compressible fluid:

A compressible fluid is a type of fluid whose volume can vary when pressure is applied, leading to corresponding changes in its density. Common examples include gases such as air, oxygen, hydrogen, and nitrogen.

2.2.4 Incompressible fluid:

An incompressible liquid ($\rho = \text{constant}$) is a fluid in which the particles remain at a fixed distance from each other, so its volume and density do not change with variations in pressure.

Typical examples of incompressible liquids include water, oil, and mercury.

Chapter 2: Reminders and definition

2.3 Physical properties of fluids:

The properties of fluids play a crucial role in both their behavior and their applications in engineering and technology. Below are some of the most important fundamental properties of liquids.

2.3.1 Density equation:

Density is the ratio between mass and the volume occupied.

$$\rho = m / V$$

ρ : Density in kg/m³

m: Mass in kg

V: Volume in m³

2.3.2 Viscosity:

A quantity that describes a fluid's ability to flow when exposed to a force. There are two main types of viscosity:

2.3.3 Dynamic viscosity:

Dynamic viscosity is a physical property that describes how a fluid responds to applied stress. It indicates the fluid's resistance to motion or flow when an external force is applied.

$$F = \mu \cdot S \cdot \Delta V / \Delta Z$$

F: Sliding force between layers (N)

μ : Dynamic viscosity (Pa. s)

S: Contact area between two layers (m²)

ΔV : Velocity difference between two layers (m/s)

ΔZ : Distance between two layers (m)

Chapter 2: Reminders and definition

2.3.4 Kinematic viscosity:

It represents the ratio between dynamic viscosity and the density of fluid.

$$\nu = \frac{\mu}{\rho}$$

ν : Kinematic viscosity in (m²/s)

ρ : Density in (kg/m³)

μ : Dynamic viscosity in (Pa.s)

2.3.5 Specific Gravity:

The density of a liquid is typically determined relative to water, by calculating the ratio of the liquid's density to that of water. For gases, density is usually measured with respect to air, which serves as the reference fluid.

$$d = \frac{\text{the density of the fluid}}{\text{the density of a reference fluid}}$$

2.3.6 Debit:

In the piping and tanks (columns, vessels, reactors, furnaces, etc.) of an operating manufacturing unit, fluids (gases or liquids, alone or in mixtures) often circulate continuously.

The quantity of product passing through a point in a pipe during a given time is called the flow rate, and we distinguish between:

- mass flow rate, which is most often expressed in:

Chapter 2: Reminders and definition

Tons per hour (t/h),

Tons per day (t/d),

Kilograms per second (kg/s) (SI unit)

- volumetric flow rate, which is most often expressed in:

Cubic meters per hour (m³/h),

Cubic meters per day (m³/d)

Cubic meters per second (m³/s)

Liters per second (l/s)

Nm³ /h for gases (t = 0°C and P = 1 atm) [12]

2.4 Types of flows:

2.4.1 Permanent or stationary flows

A steady-state flow regime is a flow condition that is independent of time. In other words, the main flow properties such as pressure, temperature, velocity, and density remain constant and do not change with time.

2.4.2 Laminar or turbulent flow: Reynolds number

The Reynolds number is a dimensionless quantity that represents the ratio of inertial forces to viscous forces within a fluid flow. It was developed based on experiments carried out by Osborne Reynolds in 1883, where he investigated fluid motion in a straight cylindrical pipe. His work showed that the flow behavior is influenced not only by velocity, but also by viscosity, pipe diameter, and the roughness of the inner surface of the pipe. The Reynolds number is expressed as follows:

$$Re = \frac{v \cdot D}{\nu}$$

Chapter 2: Reminders and definition

V: Fluid velocity (m/s). (Velocity measured at a distance from the wall).

D: Tube inner diameter (m).

U: Kinematic viscosity of the fluid (m²/s)

If:

$Re < 2000$, the flow is laminar.

$4000 > Re$, the flow is turbulent.

$2000 < Re < 4000$, the flow is transient. (This can change from one state to another; calculations must be performed in both cases to evaluate the liquid's flow regime.)

$4000 < Re < 10^5$, the flow is smooth turbulent.[13]

2.4.3 laminar flow



Figure II.1: Fluid flow in the case of laminar flow [14]

Laminar flow is a smooth, orderly flow where fluid moves in layers with little mixing. It occurs at low Reynolds numbers due to dominant viscous forces, and its velocity increases gradually from the wall to the free stream.

2.4.4 Chapter 2: Reminders and definition

2.4.5 Turbulent regime:

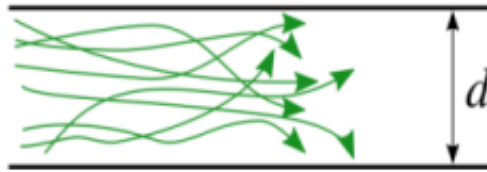


Figure II.2: Fluid stream in the case of turbulent flow [14]

Turbulent flow is a chaotic and irregular flow with strong fluctuations in velocity and pressure. It occurs at high Reynolds numbers where inertial forces dominate viscous effects. Because it is highly unpredictable, engineers study it using statistical methods rather than exact descriptions.

2.5 Concept of pressure losses:

Pressure losses happen when a fluid flows through a system like ducts or pipes and meets resistance. This resistance is mainly caused by fluid viscosity and friction with the rough inner surfaces of the walls. These losses can appear in different forms, depending on factors such as wall friction and changes in the cross-sectional area of the duct.

$$\Delta p = \Delta p_{\text{linear}} + \Delta p_{\text{singular}}$$

2.6 Linear pressure loss

Linear or distributed pressure loss is an energy loss due to friction in a tube of fixed cross-section and a certain length.

The linear pressure loss is expressed as:

$$\Delta P_{\text{linear}} = (L \cdot \lambda \cdot \rho \cdot V^2) / (2 \cdot d)$$

With:

λ = Linear pressure drop coefficient

Chapter 2: Reminders and definition

ρ = Density of the fluid in [kg/m³].

V = rate of flow in [m/s].

D = Diameter in [m]

L =Length of the pipe [m] [15]

Depending on the Reynolds number, we distinguish different cases:

Case 1: $Re < 2000$ Laminar flow, λ depends only on the Reynolds number

$$\lambda = \frac{64}{Re}$$

Case 2: $Re > 4000$ Turbulent flow

For $4000 < Re < 10^5$, the flow is said to be hydraulically smooth turbulent. λ depends only on the Reynolds number.

$$\lambda = 0.316 Re^{-0.25}$$

For $Re > 10^5$, the flow is hydraulically rough turbulent. λ depends only on λ/d

$$\lambda = 0.79 \sqrt{\frac{\varepsilon}{d}}$$

Case 3: $2000 < Re < 4000$, this is the transitional regime between laminar and turbulent flow, based on the Reynolds number and the roughness.

Chapter 2: Reminders and definition

2.7 Singular pressure loss

Singular pressure drops due to the different singularities of the network (air inlets and outlets, elbows, reductions, etc.).

The total pressure drop is the sum of the linear and singular drops. [16]

$$\Delta P_{\text{singular}} = 0.5 \cdot \rho \cdot K \cdot V^2$$

with

λ = Linear pressure drop coefficient

ρ = density of the fluid in [kg/m³]

V = rate of flow in [m/s]

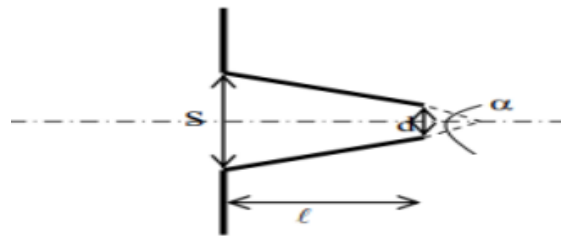
K = coefficient depend on the nature of local resistance

Chapter 2: Reminders and definition

2.8 Types of Nozzles

2.8.1 Convergent Nozzles

A convergent nozzle increases fluid velocity by narrowing its cross-section toward a throat. This design forces the fluid to speed up, making it ideal for applications requiring high velocity and momentum. It is most effective with subsonic flows, where the fluid reaches maximum speed at the throat due to a drop in pressure. Convergent nozzles are widely used in aerospace and automotive fields, such as in jet engines and fuel injection systems.[17]



FigureII.3: Converging conical nozzle

$$Q = \mu S \sqrt{2gh}$$

α : is the angle at the apex of the cone.

l : Nozzle length.

S : Orifice cross-sectional area (usually the nozzle exit area $S = \frac{\pi d^2}{4}$)

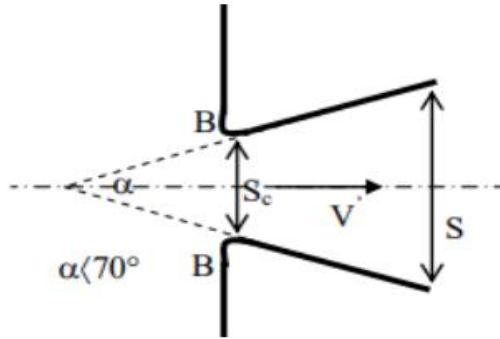
H : charge.

μ : (0,75 ÷ 0,97)

Chapter 2: Reminders and definition

2.8.2 Divergent Nozzles

A divergent nozzle has an expanding cross-section that converts high pressure and low velocity into lower pressure and higher velocity. It is commonly used in subsonic gas flow applications and is widely applied in industries such as aerospace, manufacturing, and power generation, especially in systems like rocket engines, steam turbines, and hydraulic systems.[17]



FigureII.4 : Ajutage conique divergent

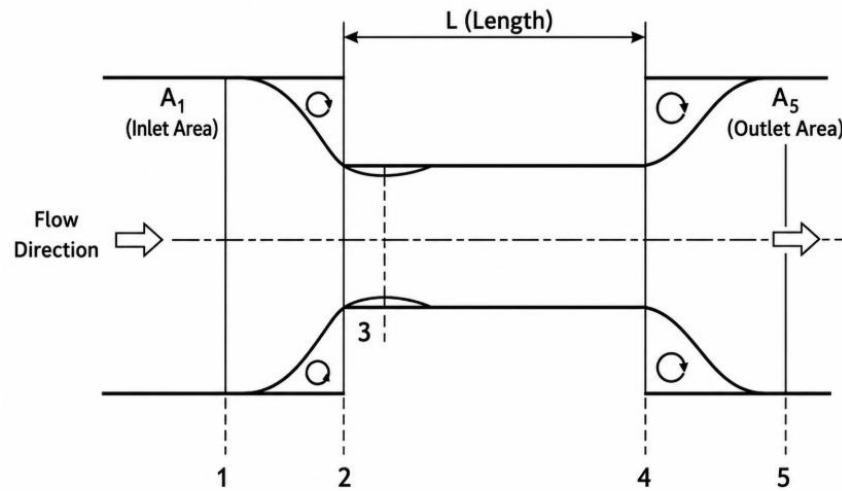
$$Q = S\sqrt{2gh}$$

2.8.3 Convergent-Divergent Nozzles

A convergent-divergent nozzle is designed to efficiently control fluid flow in both subsonic and supersonic conditions. It consists of three parts: a converging section that increases velocity, a throat where velocity is maximum, and a diverging section that adjusts velocity and pressure.

This specific type of nozzle is the focal point of our current study, where we closely investigate the dynamic effects of velocity and pressure variations along the flow path. Furthermore, our analysis places particular emphasis on the formation of recirculation zones and flow separation behaviors, examining how these phenomena influence the nozzle's overall performance. This type of nozzle is widely used in applications like rocket engines, steam turbines, and jet propulsion systems to optimize performance and efficiency...[18]

Chapter 2: Reminders and definition



FigureII.5 : Convergent-Divergent Nozzles

2.9 Conclusion

A nozzle controls how a fluid moves by changing its velocity and pressure. When fluid flows through a nozzle, a decrease in area increases velocity and reduces pressure, while an increase in area can reduce velocity and raise pressure (depending on flow conditions) This interaction allows the nozzle to convert pressure energy into kinetic energy, making the fluid flow faster and more efficiently.

III. CHAPTER 3 : Simulation, results, and interpretation.

Chapter 3: Simulation, results, and interpretation

3.1 Introduction:

One of the main issues in pipelines is the flow through an orifice, which can lead to changes in the flow system.

This chapter analyzes the behavior of nozzles in a pipe.

We also perform two-dimensional numerical simulations using finite element software, allowing us to evaluate several physical quantities related to the flow, such as the velocity field, concentration field, and pressure field and Recirculation Zones.

In this study, we focus particularly on the effects of variations in the Reynolds number, nozzle length, and pipe ratio on the flow behavior.

3.2 The model's geometry has certain conditions and limits:

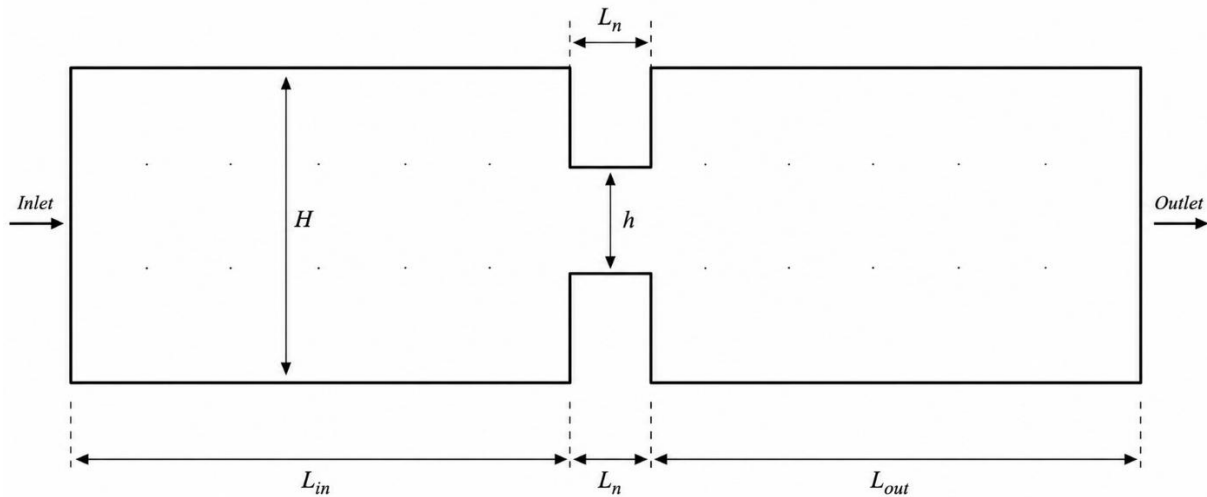


Figure III.1: Representation of the problem geometry

with boundary conditions

Chapter 3: Simulation, results, and interpretation

3.3 Hypotheses

In our simulation, we relied on the following assumptions:

- Steady flow: $1/l_t = 0$
- Incompressible fluid: $\rho = \text{constant}$
- Two-dimensional flow: $1/l_z = 0$
- Negligible gravitational forces: $\rho f = 0$
- Newtonien fluide: $\mu = \text{constant}$

3.4 Equations of Motion (Incompressible Navier–Stokes Equations)

1. Momentum Equation:

$$\rho \frac{l\vec{u}}{l_t} = \rho \vec{f} - \nabla p + \mu \nabla^2 \vec{u} \quad (III.1)$$

$$\rho \frac{l\vec{u}}{l_t} = \rho \left(\frac{l\vec{u}}{l_t} + \vec{u} \cdot \text{grad } \vec{u} \right) \quad (III.2)$$

Substituting equation (III.1) into equation (III.2), we obtain:

$$\rho \left(\frac{l\vec{u}}{l_t} + \vec{u} \cdot \text{grad } \vec{u} \right) = \rho \vec{f} - \nabla p + \mu \nabla^2 \vec{u} \quad (III.3)$$

With:

$$\begin{aligned} \frac{l\vec{u}}{l_t} &= 0 & ; & & \rho \vec{f} &= 0 \\ \rho(\vec{u} \cdot \text{grad } \vec{u}) &= -\nabla p + \mu \nabla^2 \vec{u} \\ -\mu \nabla^2 + \rho(\vec{u} \cdot \nabla) \vec{u} + \nabla p &= 0 \end{aligned} \quad (III.4)$$

2. Continuity Equation:

$$\frac{\partial \rho}{\partial t} + \text{div}(\rho \vec{u}) = 0 \quad (III.5)$$

Chapter 3: Simulation, results, and interpretation

3.5 Simulation:

3.5.1 Comsol Multiphysics:

COMSOL Multiphysics is a numerical simulation software based on the finite element method. This software allows the simulation of numerous physical phenomena and engineering applications, and particularly coupled phenomena or multiphysics simulations.

3.5.2 Mesh Convergence Study:

During simulations with finite element software, a finer mesh generally produces a more accurate solution. However, this finer mesh increases computation time.

This is why we conducted a mesh convergence study.

But mesh refinement will increase the number of mesh elements and the computation time, which will raise the issue of our computer's capacity to perform the calculations.

3.5.3 First meshing method: Extremely fine triangular:

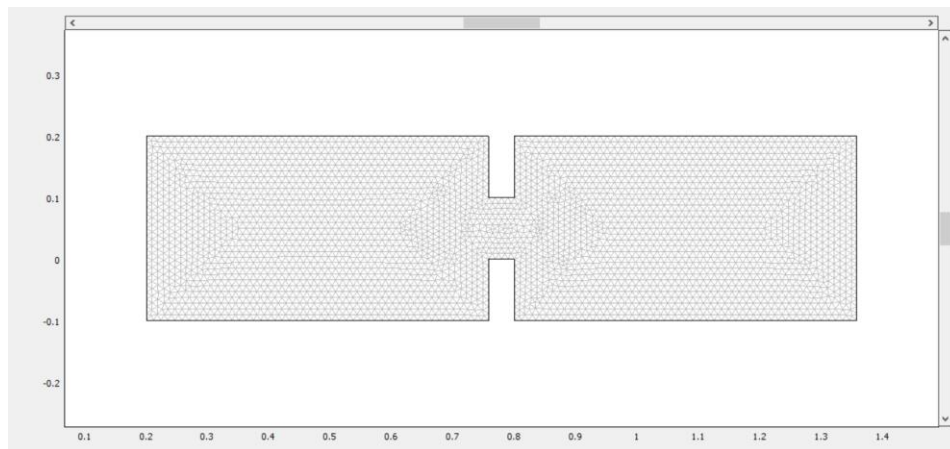


Figure III.2: Representation of the mesh

with an extremely fine triangular

Chapter 3: Simulation, results, and interpretation

This first mesh, for a Reynolds number of 500 and an expansion ratio of 3, contains 65124 elements. The computation time is 4.562 seconds. The solution converges in 6 iterations, as

shown in the following figure:

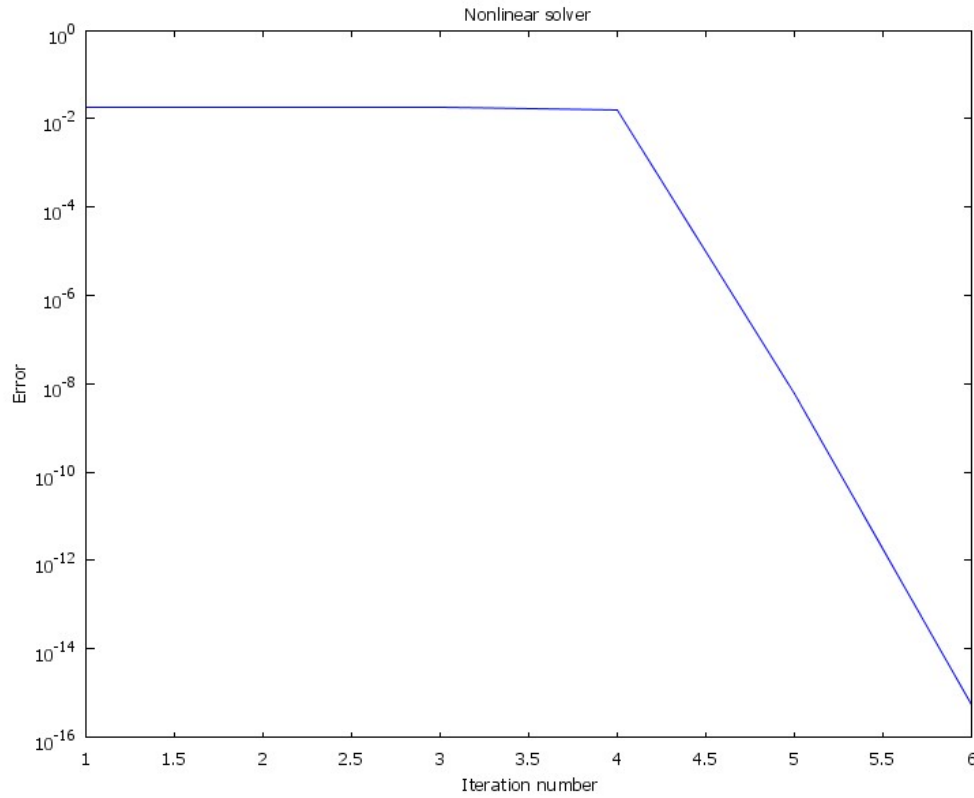


Figure III.3: Convergence of the calculation as a function of the number of iterations

3.5.4 Second meshing method: extra fine triangle:

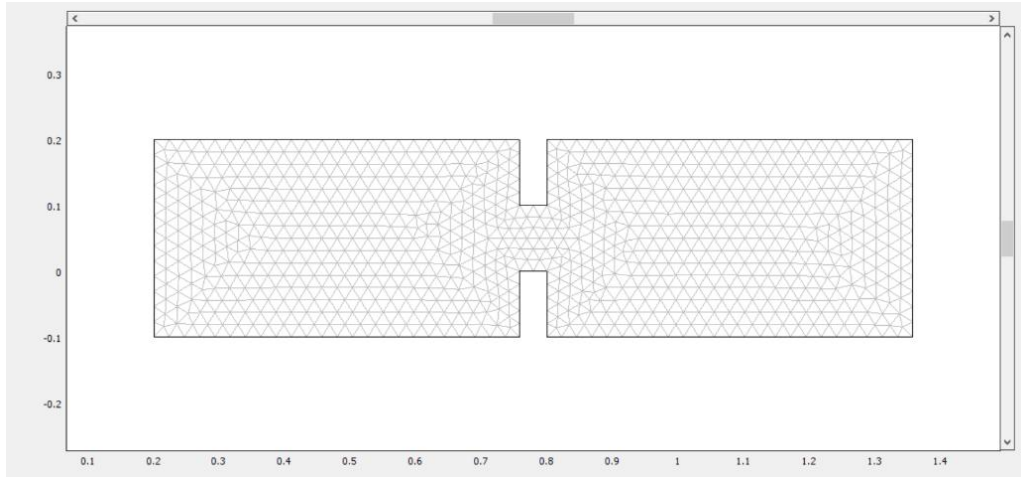


Figure III.4: Representation of the mesh with an extra fine triangle

Chapter 3: Simulation, results, and interpretation

This mesh, for a Reynolds number of 500 and an expansion ratio of 3, comprises 1670 elements. The computation time is 1.473 seconds. The solution converges in 6 iterations, as shown in the following:

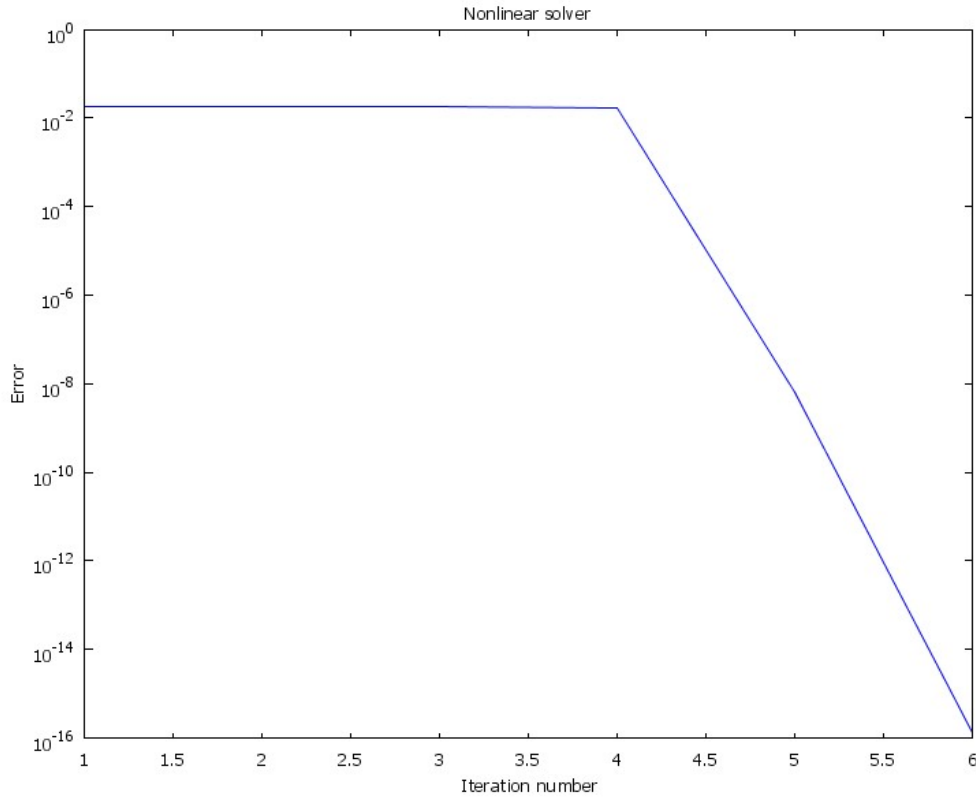


Figure III.5: Convergence of the calculation as a function of the number of iterations

3.5.5 Third meshing method: Extremely fine quadrangle

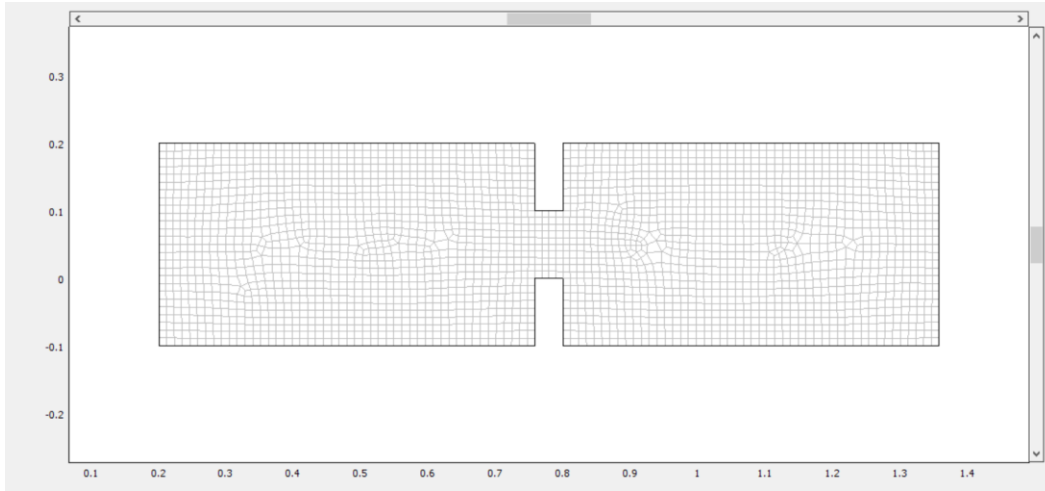


Figure III.6: Representation of the mesh with an extremely quadrangle mesh

This last mesh, for a Reynolds number of 500 and an expansion ratio of 3, contains 2678 elements. The computation time is 4.374 seconds. The solution converges after 6 iterations, as shown in the following:

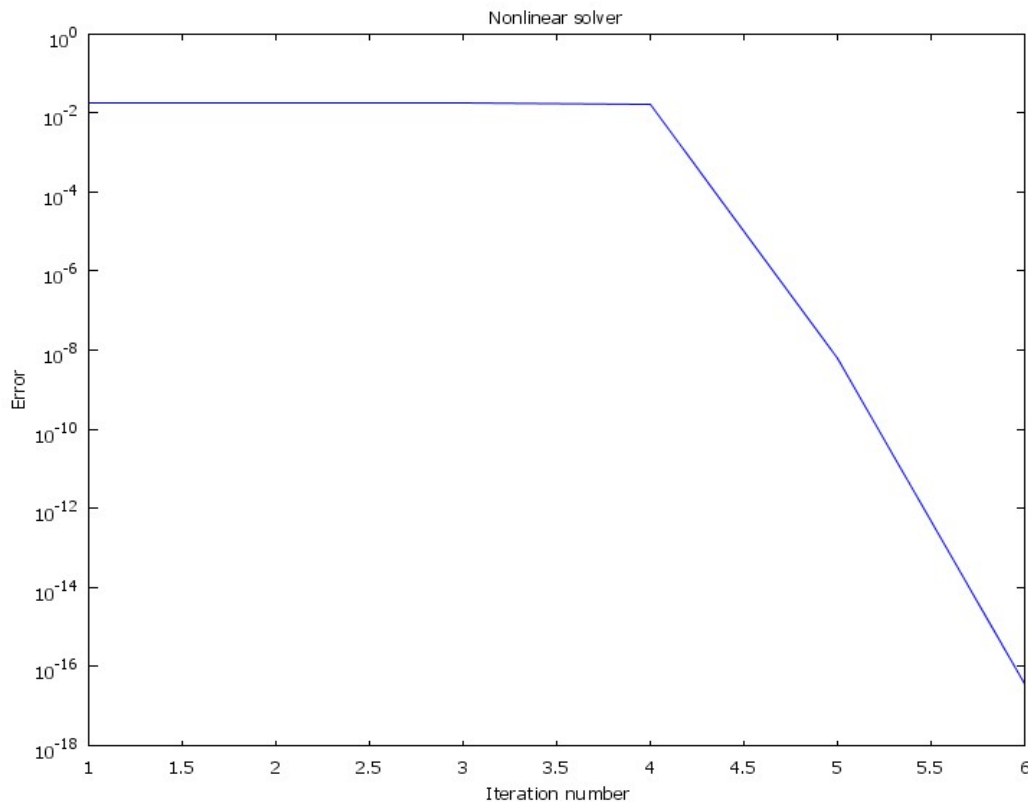


Figure III.7: Convergence of the calculation as a function of the number of iterations

Chapter 3: Simulation, results, and interpretation

The three meshes converge and give us a result, and each mesh requires a certain amount of computation time.

Based on the obtained results, the extremely fine triangular meshing method will be adopted for the remainder of the study because it offers the best compromise between accuracy and computational cost. It provides satisfactory convergence with a relatively short computation time, making it the most efficient and reliable choice for the simulations.

3.6 Results and discussion

As mentioned previously, we will study the flow of water through a pipe gradually widening with different Reynolds numbers (500; 1000; 1500; 2000; 2500), different expansion ratios (3; 4; 5; 6; 7) and the influence of expansion length (0.042 ; 0.084 ; 0.126 ; 0.168 ; 0.21)

3.7 Study of the influence of the Reynolds number

In this section, the influence of the Reynolds number on the flow behavior inside the convergent-divergent nozzle is investigated. The nozzle length is kept constant at $l_n=0.042\text{m}$, while the enlargement ratio is fixed at $r=3$. The Reynolds number is varied to analyze its effect on velocity, pressure distribution, recirculation zone

3.8 Calculation of the Reynolds number

To be able to vary the Reynolds number, it is necessary to find the speed corresponding to each desired Reynolds number.

Here is an example of the calculation for the water:

$$\text{Re}=500$$

$$\rho=997 \text{ Kg/m}^3$$

$$\mu=8.91 \cdot 10^{-5} \text{ Pa}\cdot\text{s}$$

$$D = 0.3\text{m}$$

$$\text{Re} = \frac{\rho \cdot V \cdot D}{\mu} \longrightarrow V = \frac{\text{Re} \cdot \mu}{\rho \cdot D}$$

Chapter 3: Simulation, results, and interpretation

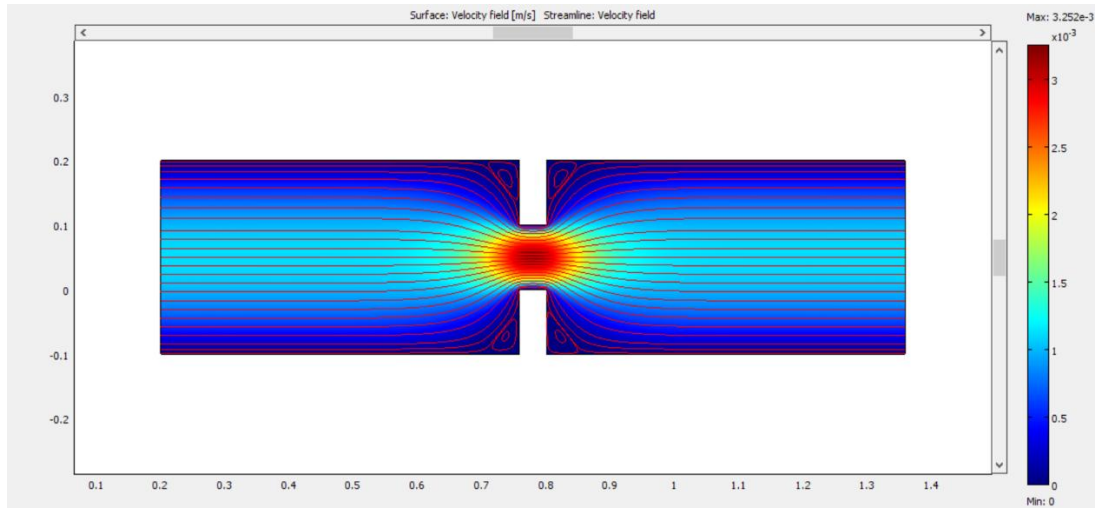


Figure III.8: Streamlines for a Reynolds number of 500

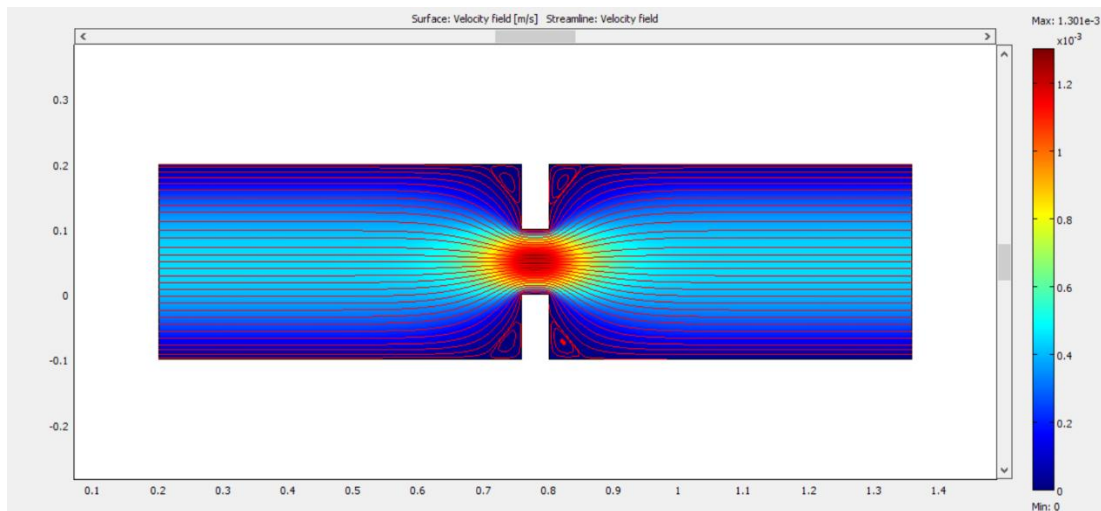


Figure III.9: Streamlines for a Reynolds number of 1000

Chapter 3: Simulation, results, and interpretation

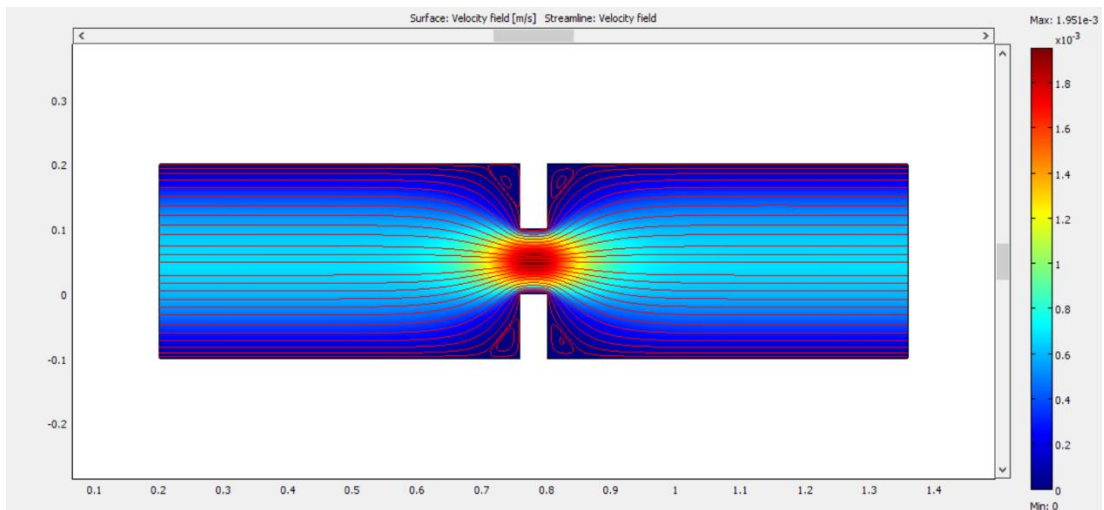


Figure III.10: Streamlines for a Reynolds number of 1500

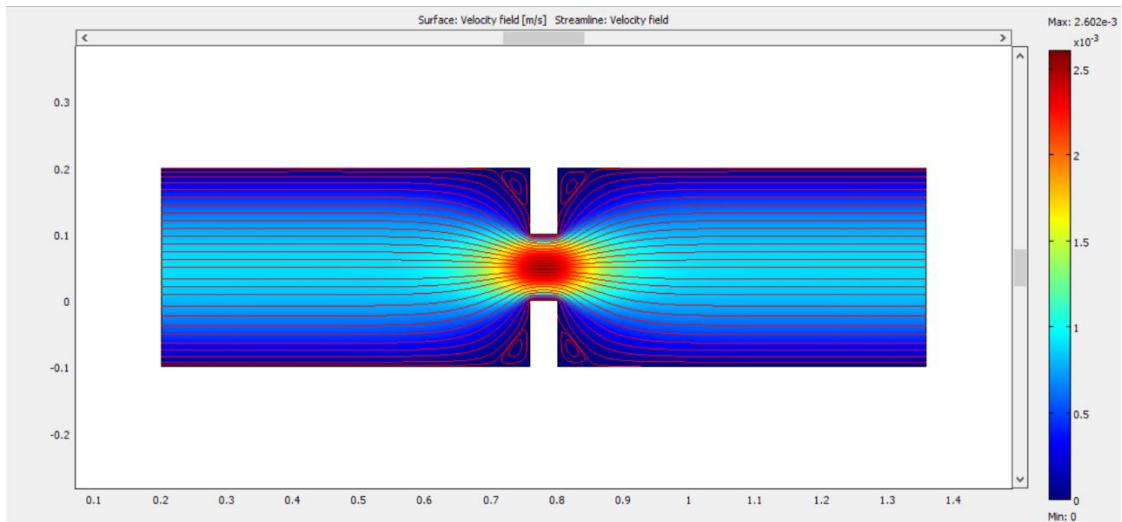


Figure III.11: Streamlines for a Reynolds number of 2000

Chapter 3: Simulation, results, and interpretation

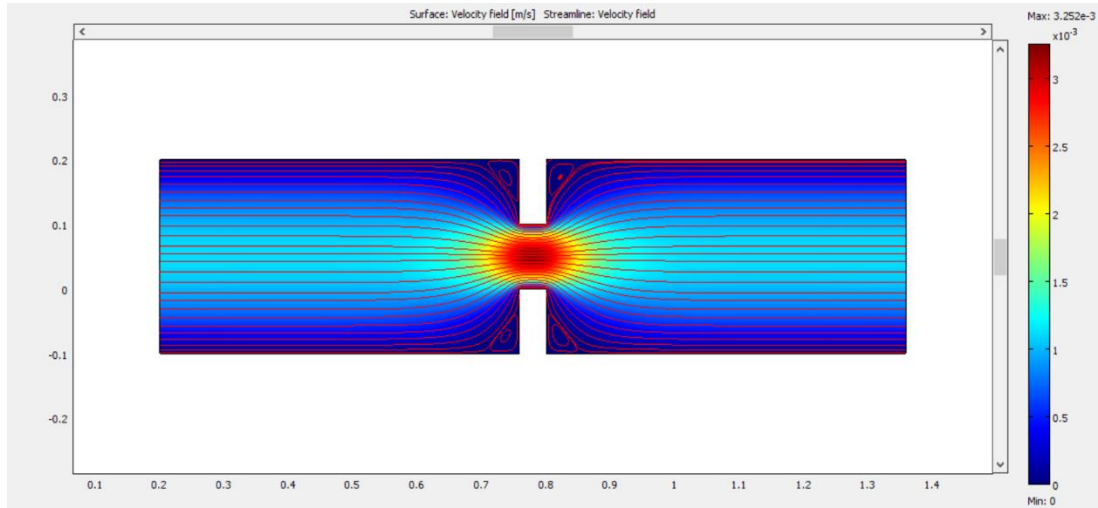


Figure III.12: Streamlines for a Reynolds number of 2500

The figures III.1, III.2, III.3, III.4 and III.5 representant the streamlines in the function Different types of Reynolds numbers.

The flow behavior corresponding to Reynolds numbers of 500, 1000, 1500, 2000, and 2500 demonstrates a pronounced acceleration through the constricted section due to the reduction in the flow area. The velocity reaches its maximum value at the throat region, where the flow is subjected to the strongest confinement. As the Reynolds number increases, inertial effects become increasingly significant, resulting in higher velocity magnitudes and steeper velocity gradients. Downstream of the constriction, the sudden expansion induces an adverse pressure gradient that promotes flow separation and the formation of a recirculation zone. The length and intensity of this recirculation region increase progressively with Reynolds number, reflecting the enhanced dominance of inertial forces over viscous effects. Consequently, the flow reattachment point is displaced further downstream, accompanied by the development of larger vortical structures within the separated flow region. These findings highlight the critical influence of Reynolds number on the mechanisms of flow separation, recirculation, and momentum transport in constricted channel flows.

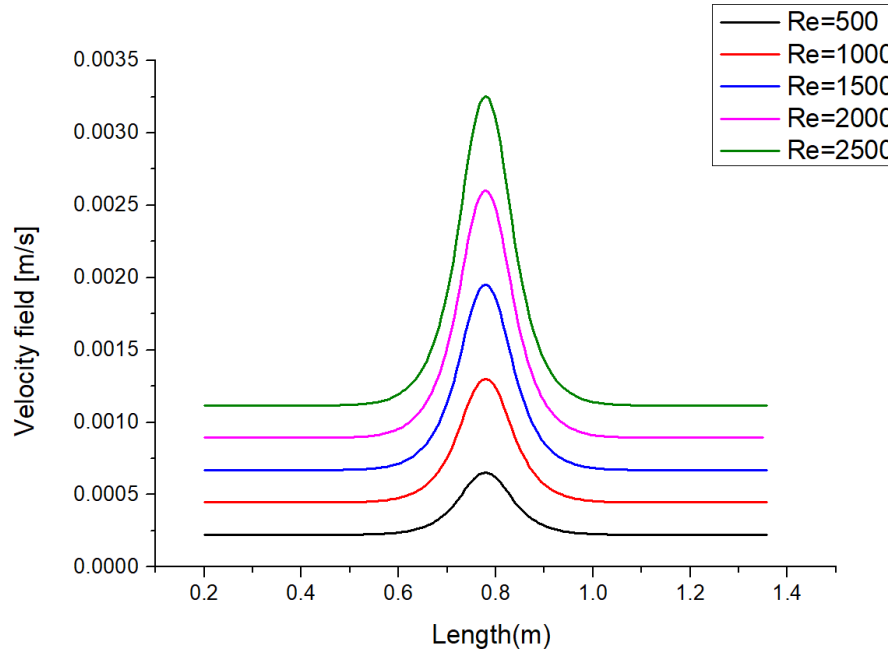


Figure III.13: Velocity field along the axis of symmetry of the pipe for different Reynolds numbers

The velocity distribution clearly demonstrates the influence of the Reynolds number on the flow through the nozzle. As the Reynolds number increases from 500 to 2500, the fluid velocity rises significantly throughout the flow domain. The highest velocities are observed in the throat region, where the cross-sectional area reaches its minimum value. This acceleration is a direct consequence of the continuity principle, which requires the fluid to increase its velocity as the flow area decreases. Furthermore, higher Reynolds numbers lead to stronger velocity gradients and greater flow momentum downstream of the nozzle. The results indicate that increasing the Reynolds number enhances the inertial effects of the flow, resulting in a more pronounced velocity field and improved fluid transport characteristics.

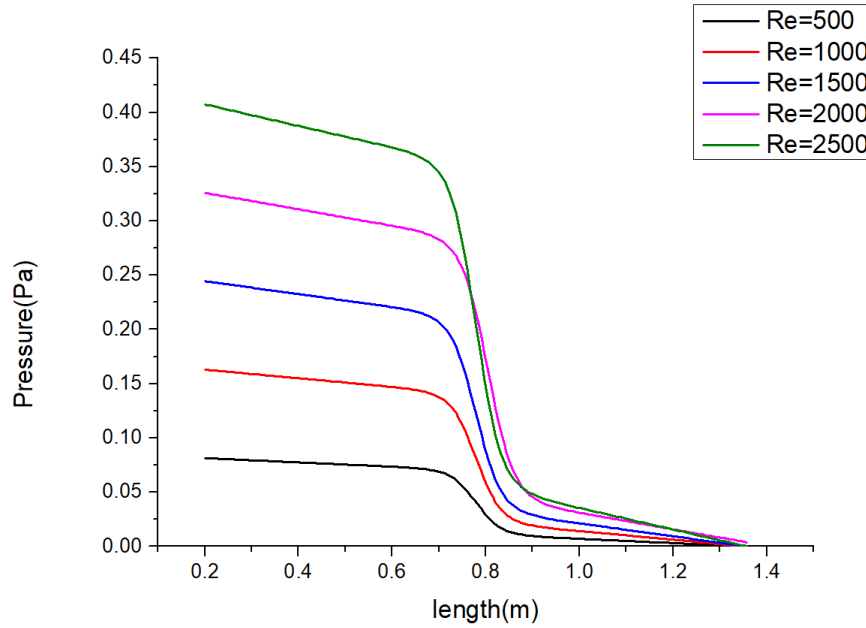


Figure III.14: pressure field along the axis of symmetry of the pipe for different Reynolds numbers

The pressure distribution exhibits an inverse relationship with the velocity field and highlights the effect of the Reynolds number on pressure variations within the nozzle. As the fluid approaches the throat section, the pressure decreases sharply due to the conversion of pressure energy into kinetic energy. This pressure reduction becomes increasingly pronounced as the Reynolds number increases from 500 to 2500. At higher Reynolds numbers, the fluid experiences stronger acceleration, leading to larger pressure gradients and a greater overall pressure drop across the nozzle. These findings are consistent with Bernoulli's principle and demonstrate that increasing the Reynolds number intensifies the pressure variations within the flow domain, particularly in regions of high acceleration.

3.9 Study of the influence of length (L_n):

In this section, the effect of nozzle length on the flow behavior is investigated. The Reynolds number is kept constant at $Re = 500$ and the enlargement ratio is fixed at $r = 3$. Different nozzle lengths, ranging from 0.042 m to 0.252 m, are considered to analyze their influence on the velocity field, pressure distribution, and recirculation zones within the nozzle.

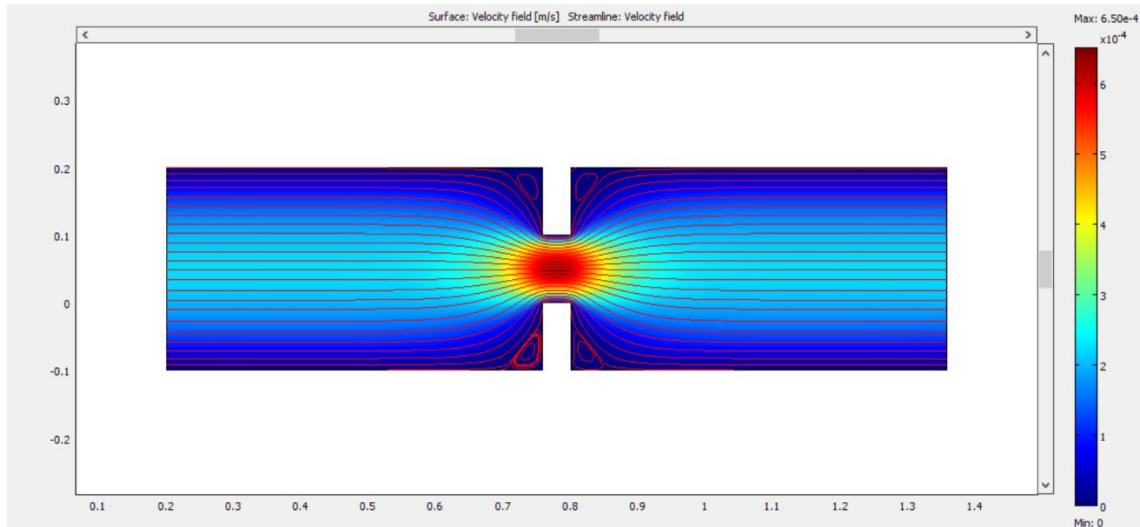


Figure III.15 Streamlines for a length $l_n = 0.042$ m

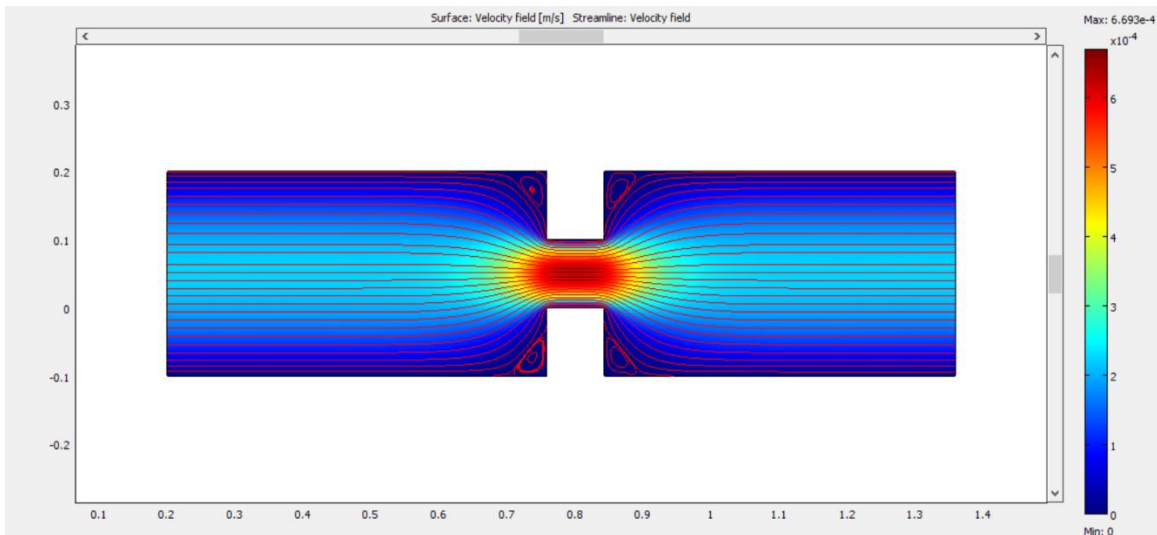


Figure III.16 Streamlines for a length $l_n = 0.084$ m

Chapter 3: Simulation, results, and interpretation

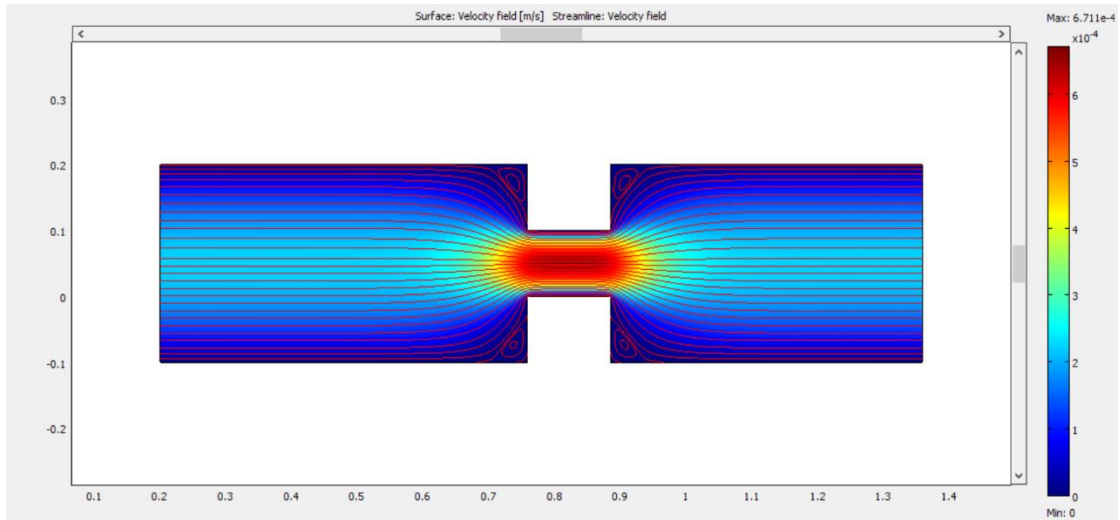


Figure III.17 Streamlines for a length $l_n=0.126\text{m}$

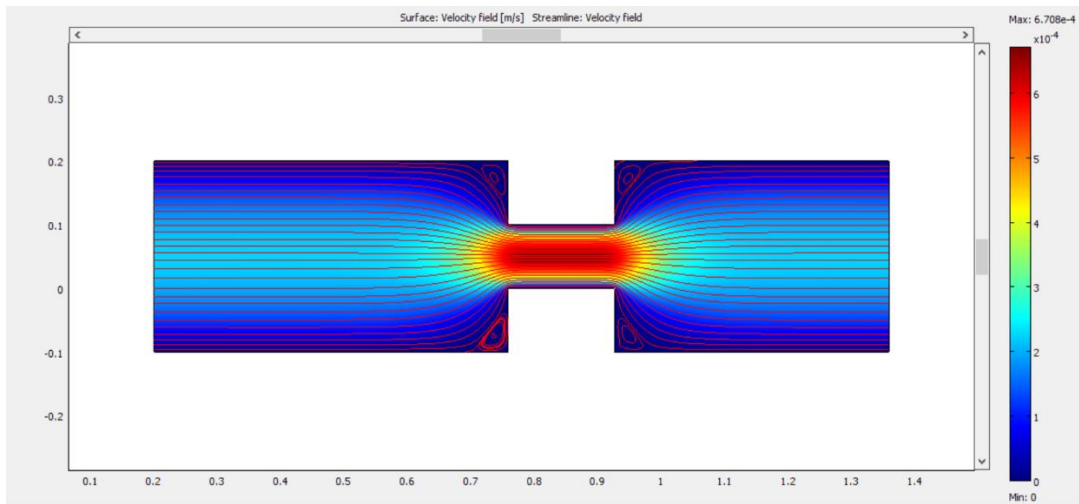


Figure III.18 Streamlines for a length $l_n=0.168\text{m}$

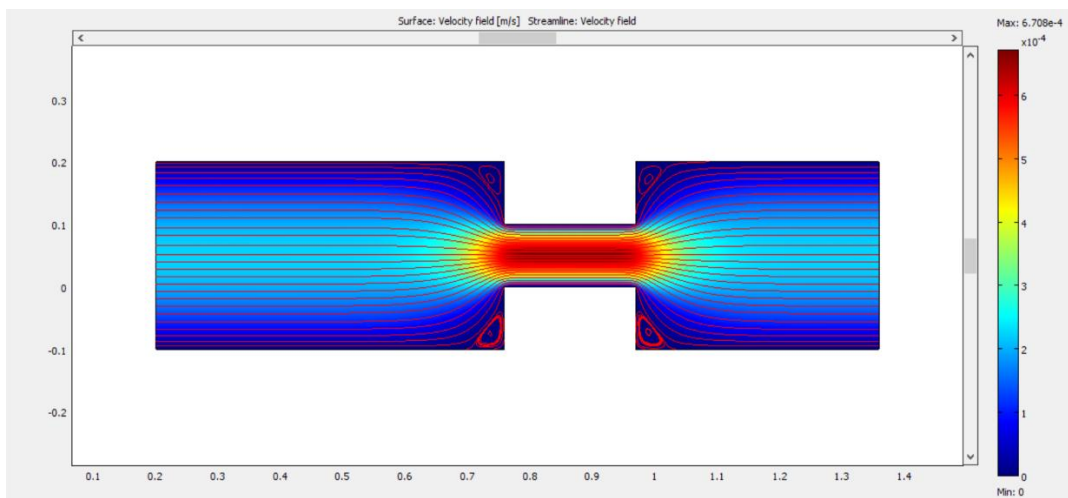


Figure III.19 Streamlines for a length $l_n=0.21\text{m}$

Chapter 3: Simulation, results, and interpretation

The figures III.6, III.7, III.8, III.9 and III.10 representant the streamlines in the function
Different types of the influence of length

he flows behavior corresponding to nozzle lengths of $L_n=0.042, 0.084, 0.126, 0.168$, and 0.210 reveals a significant influence of the nozzle geometry on the downstream recirculation characteristics. As the nozzle length increases, the fluid experiences a more gradual acceleration within the converging section, leading to a modification of the velocity and pressure fields at the nozzle exit. Following the sudden expansion, flow separation occurs and a recirculation zone develops immediately downstream of the constriction

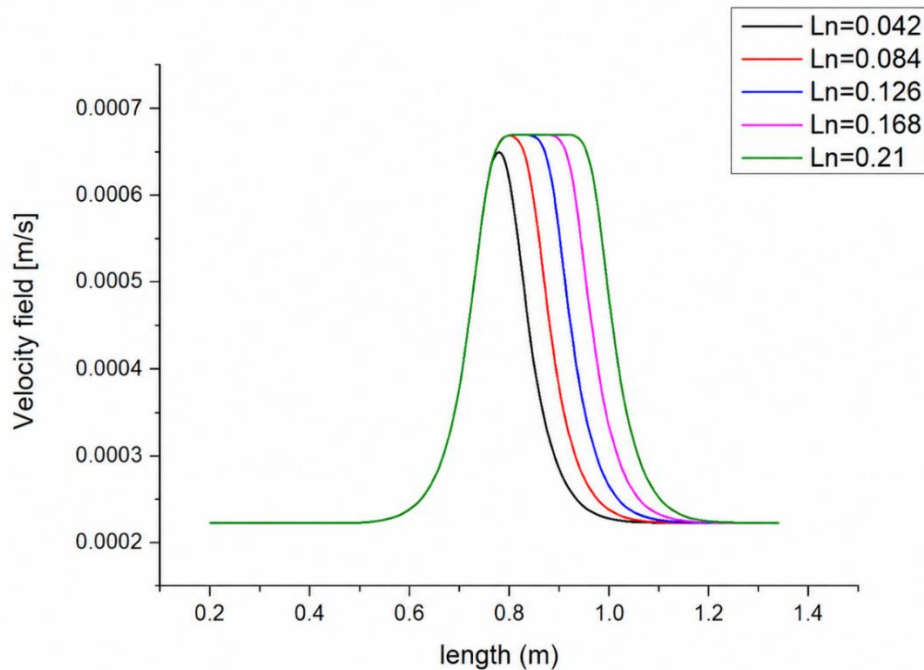


Figure III.20: Velocity field along the axis of symmetry of the pipe for the influence of length

Chapter 3: Simulation, results, and interpretation

The axial velocity distribution for different nozzle lengths ($L_n = 0.042\text{--}0.21\text{ m}$). The results indicate that increasing the nozzle length shifts the high-velocity region downstream and extends the acceleration zone within the converging section. While the maximum velocity remains nearly constant at approximately $6.6 \times 10^{-4}\text{ m/s}$ for all cases, the location over which this peak velocity is maintained becomes wider as the nozzle length increases. For $L_n = 0.21\text{ m}$, the velocity profile exhibits the smoothest acceleration and deceleration behavior, indicating a more gradual flow development. In contrast, the shortest nozzle ($L_n = 0.042\text{ m}$) produces a sharper velocity gradient near the throat and a more abrupt decrease downstream. These observations demonstrate that nozzle length mainly affects the spatial distribution of velocity rather than its maximum magnitude.

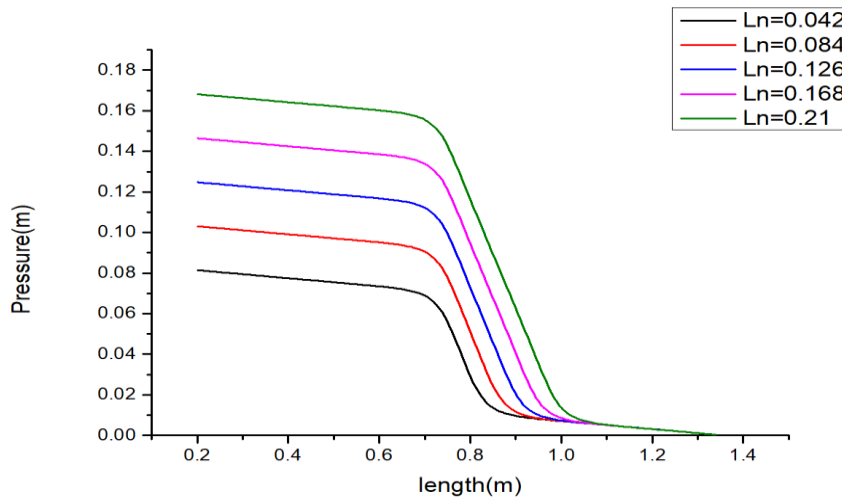


Figure III.21: pressure field along the axis of symmetry of the pipe for the influence of length

the pressure distribution along the channel for different nozzle lengths. As the nozzle length increases from 0.042 m to 0.21 m, the pressure drop is distributed over a longer distance, resulting in smoother pressure gradients within the converging section. The highest-pressure values are observed upstream of the nozzle and increase with nozzle length, reaching approximately 0.17 for $L_n = 0.21\text{ m}$, compared with about 0.08 for $L_n = 0.042\text{ m}$. A rapid pressure decrease occurs near the throat region due to flow acceleration, after which the pressure approaches nearly zero downstream. The longer nozzles exhibit a more gradual pressure reduction before the throat and a delayed pressure recovery, reflecting the stronger influence of nozzle geometry on the momentum exchange and flow development.

Chapter 3: Simulation, results, and interpretation

3.10 Study of the influence of the expansion report r :

In this section, the influence of the expansion ratio (r) on the flow characteristics is investigated. The nozzle length ($L_n = 0.042$ m) and the Reynolds number ($Re = 500$) are kept constant, while the expansion ratio is varied from 3 to 7. The upstream channel height is fixed at $D_1 = 0.30$ m, whereas different downstream channel heights (D_2) are considered to achieve the desired expansion ratios. The objective is to evaluate the effect of the expansion ratio on the velocity field, pressure distribution, recirculation zones.

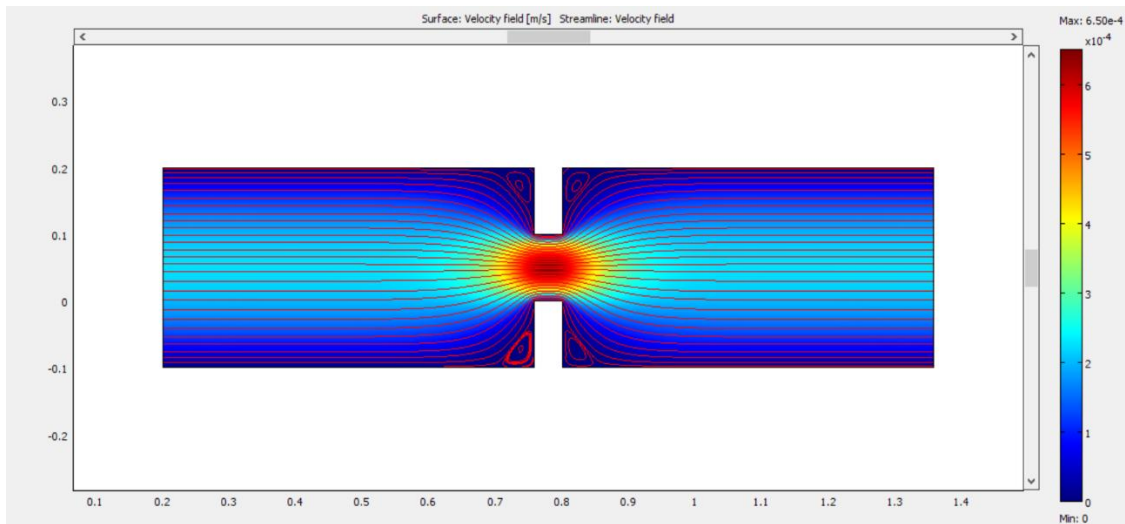


Figure III .22 Streamlines for the expansion report $r=3$

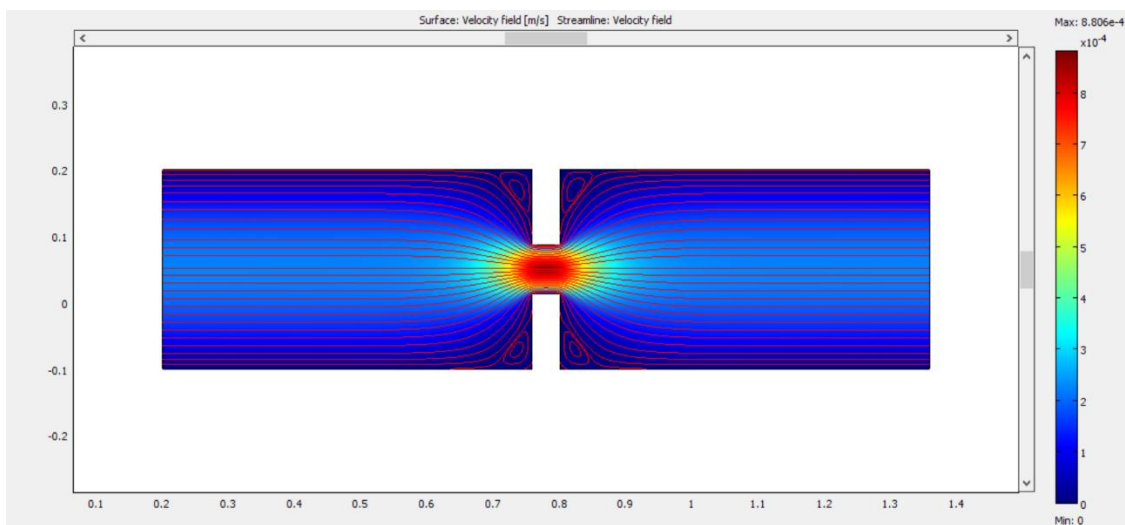


Figure III.23 Streamlines for the expansion report $r=4$

Chapter 3: Simulation, results, and interpretation

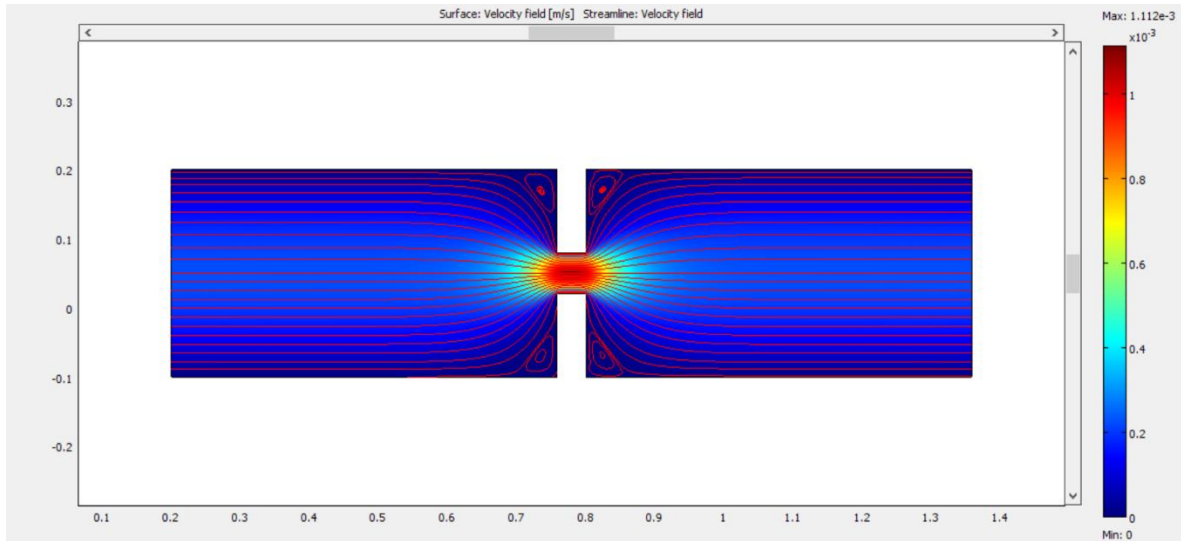


Figure III.24 Streamlines for the expansion report $r=5$

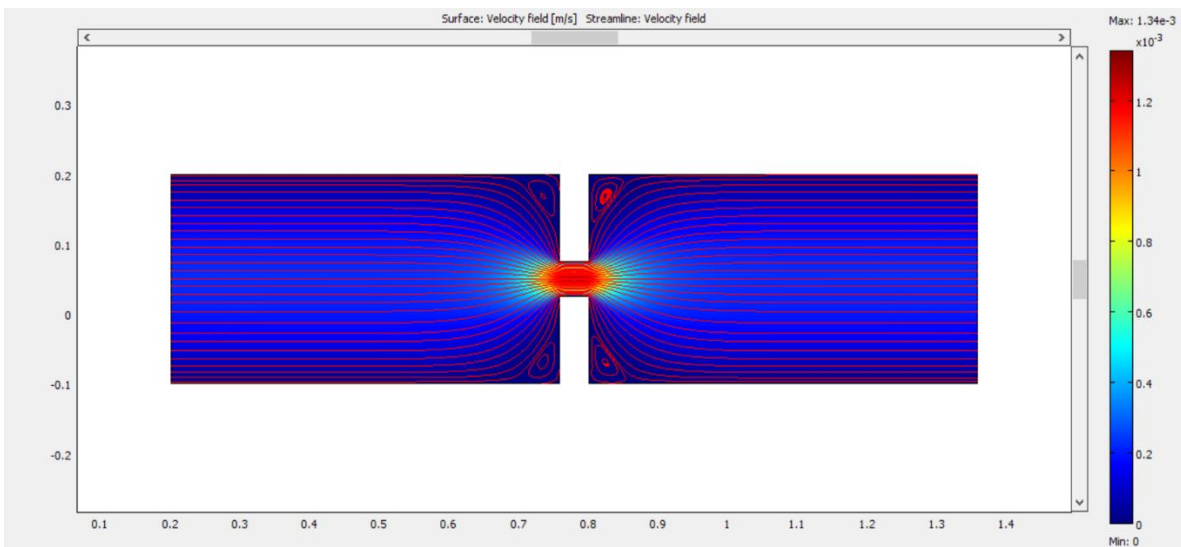


Figure III.25 Streamlines for the expansion report $r=6$

Chapter 3: Simulation, results, and interpretation

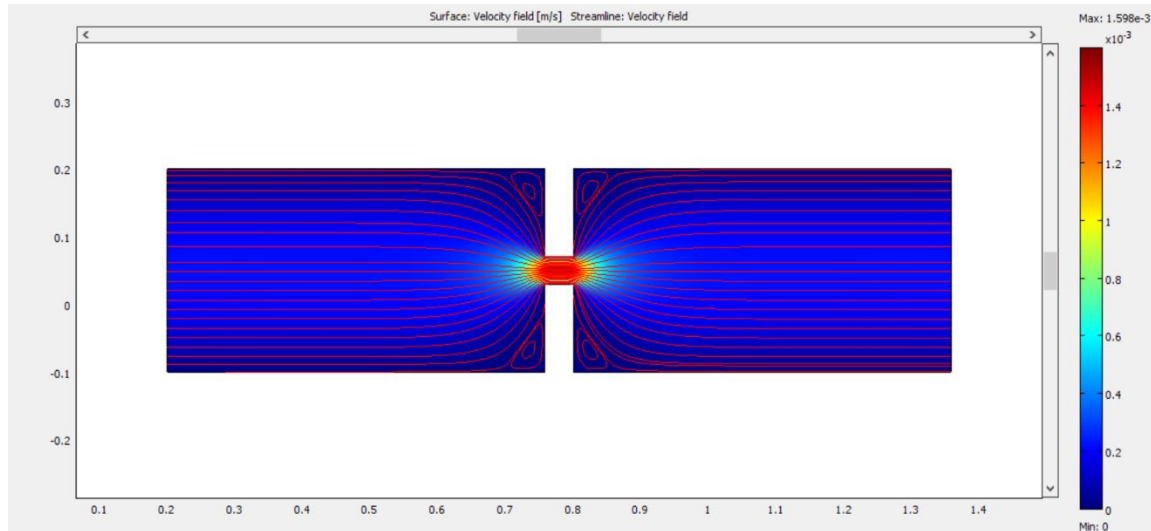


Figure III.26 Streamlines for the expansion report $r=7$

**The figures III.11, III.12, III.13, III.14 and III.15 representant the streamlines in the function
Different types of the expansion report**

the flow behavior corresponding to expansion ratios of $R = 3, 4, 5, 6$, and 7 reveals a significant influence of the nozzle geometry on the velocity distribution and downstream flow structure. As the expansion ratio increases, the reduction in the throat diameter causes a stronger acceleration of the fluid, resulting in higher maximum velocities at the throat and a more concentrated high-speed jet. The increased flow acceleration is accompanied by steeper velocity gradients in the constricted region. Downstream of the throat, the sudden expansion generates an adverse pressure gradient that promotes flow separation and the formation of a recirculation zone. As the expansion ratio increases, both the size and intensity of the recirculation region become more pronounced, indicating stronger vortex development and enhanced momentum exchange between the main flow and the separated region. Consequently, the flow reattachment point shifts further downstream, while the vortical structures become larger and more persistent. These results demonstrate that the expansion ratio plays a critical role in controlling flow acceleration, separation, recirculation, and the overall hydrodynamic behavior within the converging-diverging nozzle.

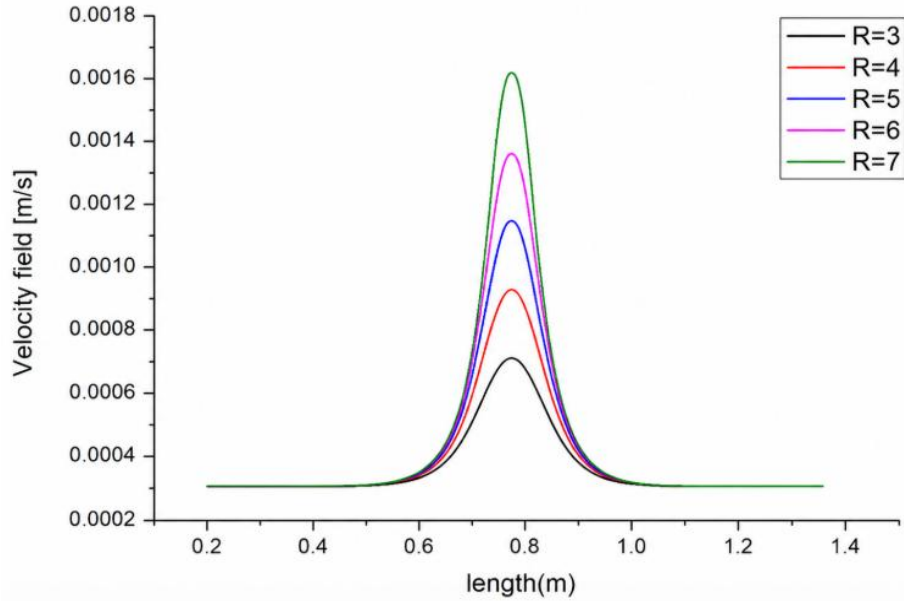


Figure III.27: Velocity field along the axis of symmetry of the pipe for different expansion ratios

The velocity distribution demonstrates a significant influence of the expansion ratio (R) on the flow acceleration within the nozzle. As the expansion ratio increases from $R = 3$ to $R = 7$, the reduction in the throat diameter causes the fluid to accelerate more intensely, resulting in progressively higher peak velocities at the throat section. The velocity profiles exhibit a sharp increase in the converging region, reaching their maximum values at the minimum flow area before decreasing downstream as the flow expands. Furthermore, higher expansion ratios produce a narrower and more concentrated high-speed jet, indicating stronger flow acceleration and enhanced momentum transport through the constricted section. These results confirm that the expansion ratio is a key geometric parameter governing the velocity field inside the nozzle.

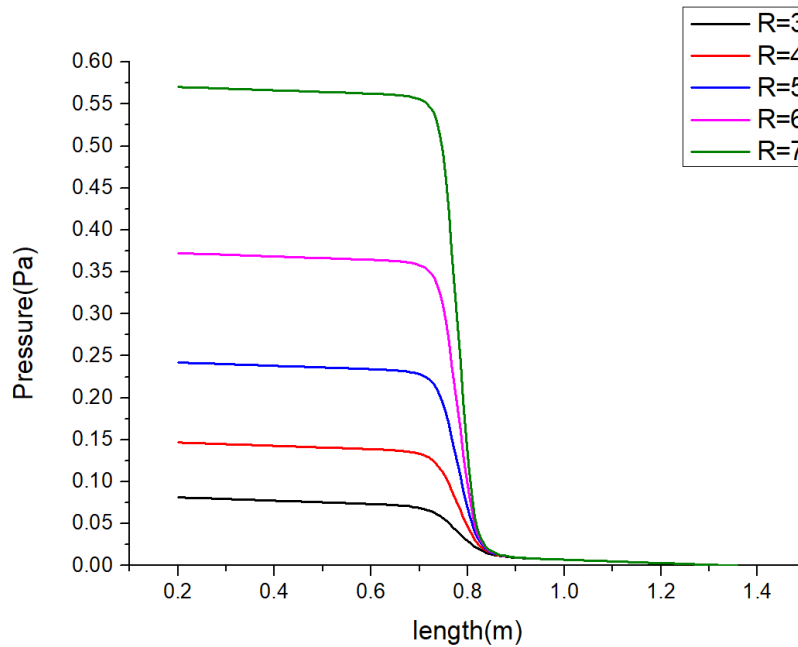


Figure III.28: pressure field along the axis of symmetry of the pipe for different expansion ratios

The pressure distribution reveals that the expansion ratio has a pronounced effect on the pressure field throughout the nozzle. As the expansion ratio increases from $R = 3$ to $R = 7$, the pressure upstream of the throat becomes progressively higher, while a sharp pressure drop occurs across the throat due to the rapid acceleration of the fluid. This pressure decrease is associated with the conversion of pressure energy into kinetic energy in accordance with Bernoulli's principle. Higher expansion ratios generate steeper pressure gradients and a more significant pressure reduction in the throat region, whereas the pressure downstream remains nearly constant after the expansion. These observations indicate that increasing the expansion ratio enhances the pressure loss across the constricted section and strongly influences the overall pressure distribution within the nozzle.

Chapter 3: Simulation, results, and interpretation

3.11 Conclusion:

This study demonstrated that the Reynolds number, nozzle length, and expansion ratio have a significant influence on flow behavior in a converging-diverging nozzle. Increasing the Reynolds number enhances fluid velocity, pressure drop, and recirculation intensity. Increasing nozzle length promotes smoother flow development and pressure variation, while a higher expansion ratio results in stronger acceleration, larger recirculation zones, and greater pressure losses. The comparison between the numerical results and Bernoulli's principle showed good agreement, as regions of high velocity correspond to low pressure and vice versa. These findings highlight the importance of both flow conditions and geometric parameters in optimizing nozzle performance.

General Conclusion

This work presented a numerical study of water flow through a nozzle using COMSOL Multiphysics. The objective was to investigate the influence of the Reynolds number, nozzle length, and expansion ratio on the velocity and pressure fields.

The results showed that the fluid accelerates significantly in the throat region, where the maximum velocity and minimum pressure are observed. Increasing the Reynolds number enhances flow momentum and increases both velocity and pressure gradients. The nozzle length affects the flow development by promoting a smoother acceleration and a more stable velocity distribution. In addition, the expansion ratio strongly influences the flow behavior; a smaller throat diameter leads to higher velocities and larger pressure drops.

Overall, the study confirms that the Reynolds number, nozzle length, and expansion ratio are key parameters controlling the hydrodynamic behavior of flow through a nozzle. The numerical simulations provided a clear understanding of the relationship between nozzle geometry and flow performance, which can be useful for the design and optimization of industrial nozzle systems

Confirmed by Philippe GLENAT [18] and The results were validated using Bernoulli's equation, through verification of the relationship between pressure and velocity within the flow. The calculations showed good agreement with Bernoulli's principle, which states that an increase in fluid velocity leads to a decrease in pressure, and vice versa. This agreement confirms the validity of the experimentally/numerically obtained results.

References

- [1] Ohta, J and al (1993) “Performance and flow characteristics of nozzles for initially subcooled hot water (influence of turbulence and decompression rate)”. *International Journal of Multiphase Flow*, 19(1), 125–136. [https://doi.org/10.1016/0301-9322\(93\)90028-S](https://doi.org/10.1016/0301-9322(93)90028-S)
- [2] Shademan, M. and al (2012). “CFD Study of Effects of Geometry Variations on Flow in a Nozzle”. *Engineering Applications of Computational Fluid Mechanics*, 6(3), 412–425. <https://doi.org/10.1080/19942060.2012.11015432>
- [3] Luginsland, T. (2015). “How the Nozzle Geometry Impacts Vortex Breakdown in Compressible Swirling-Jet Flows”. *AIAA Journal*, 53(10), 2955–2969. <https://doi.org/10.2514/1.J053843>
- [4] Liao, W., & Deng, X. (2017). “Study on Flow Field Characteristics of Nozzle Water Jet in Hydraulic Cutting”. *IOP Conference Series: Earth and Environmental Science*, 81, Art. no. 012167. <https://doi.org/10.1088/1755-1315/81/1/012167>
- [5] Zhang, T and al. (2017). “Spray characteristics of pressure-swirl nozzles at different nozzle diameters, L/D ratios and injection pressures.” *Applied Thermal Engineering*, 121, 484–494.
- [6] Ishak, M. H. H. and al. (2019). “Numerical Analysis of Nozzle Flow and Spray Characteristics from Different Nozzles Using Diesel and Biofuel Blends”. *Energies*, 12(2), 281. <https://doi.org/10.3390/en12020281>
- [7] Jiang, T and al (2022). “Effect of Nozzle Geometry on the Flow Dynamics and Resistance Inside and Outside the Cone-Straight Nozzle”. *ACS Omega*, 7(11), 9502–9515. <https://doi.org/10.1021/acsomega.1c07050>
- [8] Khan, A., et al. (2024). “Comprehensive CFD Analysis of Base Pressure Control Using Quarter Ribs in a Sudden Expansion Duct at Sonic Mach Numbers”. *International Journal of Thermofluids*, 24, Art. no 100908. <https://doi.org/10.1016/j.ijft.2024.100908>
- [9] Zhu, X., et al. (2025). “Analysis of the Influence of Nozzle Structure and Hydraulic Parameters on the Cutting Efficiency of High-Pressure Abrasive Water Jets”. *Journal of Applied Fluid Mechanics*, 18(5), 1246–1265. https://www.jafmonline.net/article_2629.html

- [10] F. M. White (1999), "Fluid Mechanics, 4th ed", Boston, MA, USA: McGraw-Hill Higher Education, 1
- [11] V. Ralston Giles et al (2000) "mécanique des fluides et hydraulique" Série Schaum 2eme Edition McGraw-Hill
- [12] . "IFP. Training, grandeurs physiques", Edition ENSPM Formation Industrie, (2005), P9
- [13] Ben Hammouda, R., et al. (2008). "Notions de mécanique des fluides : Cours et exercices corrigés". Tunis: Centre de Publication Universitaire, Chapitre 4 : Dynamique des Fluides Incompressibles Réels, Section 3 : Régimes d'écoulement – nombre de Reynolds, p. 88.
- [14] <https://www.techno-science.net/glossaire-definition/Ecoulement-laminaire.html> DATE 12/05/2026
- [15] Çengel, Y. A., & Cimbala, J. M. (2018). Fluid Mechanics: Fundamentals and Applications, 4th ed., McGraw-Hill, Chapter 8: Internal Flow.
- [16] S. Amiroudine and J.-L. Battaglia, "Mécanique des fluides : Cours et exercices corrigés". Paris, France: Dunod, 2011, Chapitre 9 : Théorème de Bernoulli généralisé.
- [17] Çengel, Y. A. and al . (2018). "Fluid Mechanics: Fundamentals and Applications (4th ed.)" . McGraw-Hill Education. Chapter 5 page 185 195
- [18] P. Glenat and al, " Pertes de pression des écoulements monophasiques et diphasiques dans des élargissements, rétrécissements et ajutages", Rapport CEA-R-5289, Commissariat à l'Énergie Atomique (CEA), Centre d'Études Nucléaires de Grenoble, France, Nov. 1984, Chapitre 2.3 « Ajutage », pp. 12–13.