

FIRAT UNIVERSITY
GRADUATE SCHOOL OF NATURAL AND APPLIED SCIENCES
TÜRKİYE



**EXPERIMENTAL AND NUMERICAL INVESTIGATION OF OPEN
CHANNEL FLOW CHARACTERISTICS OF NEWTONIAN AND
NON-NEWTONIAN FLUIDS**

Azeez Majeed MOHAMMED

Doctoral Thesis

DEPARTMENT OF MECHANICAL ENGINEERING

Division of Energy

MAY 2024

FIRAT UNIVERSITY
GRADUATE SCHOOL OF NATURAL AND APPLIED SCIENCES
T Ü R K İ Y E

Department of Mechanical Engineering

Doctoral Thesis

**EXPERIMENTAL AND NUMERICAL INVESTIGATION OF OPEN
CHANNEL FLOW CHARACTERISTICS OF NEWTONIAN AND NON-
NEWTONIAN FLUIDS**

Author

Azeez Majeed MOHAMMED

Supervisor

Prof. Dr. Nevin ÇELİK

Co-Supervisor

Assoc. Prof. Dr. Thamer Mohammed AHMED

MAY 2024

ELAZIG

FIRAT UNIVERSITY
GRADUATE SCHOOL OF NATURAL AND APPLIED SCIENCES
T Ü R K İ Y E

Department of Mechanical Engineering

Doctoral Thesis

Title:	Experimental and Numerical Investigation of Open Channel Flow Characteristics of Newtonian and Non-Newtonian Fluids
Author:	Azeez Majeed MOHAMMED
Submission Date:	10 May 2024
Defense Date:	27 May 2024

THESIS APPROVAL

This thesis, which was prepared according to the thesis writing rules of the Graduate School of Natural and Applied Sciences, Firat University, was evaluated by the committee members who have signed the following signatures and was unanimously approved after the defense exam made open to the academic audience.

Supervisor:	Prof. Dr. Nevin ÇELİK Firat University, Department of Mechanical Engineering	<i>Signature</i> Approved
Chair:	Prof. Dr. Kadir BİLEN Ataturk University, Department of Mechanical Engineering	Approved
Member:	Prof. Dr. Ertan BUYRUK Cumhuriyet University, Department of Mechanical Engineering	Approved
Member:	Prof. Dr. İhsan DAĞTEKİN Firat University, Department of Mechanical Engineering	Approved
Member:	Prof. Dr. Mete Onur KAMAN Firat University, Department of Mechanical Engineering	Approved

This thesis was approved by the Administrative Board of the Graduate School on

..... / / 20

Signature

Prof. Dr. Burhan ERGEN
Director of the Graduate School

DECLARATION

I hereby declare that I wrote this Doctoral Thesis titled “Experimental and Numerical Investigation of Open Channel Flow Characteristics of Newtonian and Non-Newtonian Fluids ” in consistent with the thesis writing guide of the Graduate School of Natural and Applied Sciences, Firat University. I also declare that all information in it is correct, that I acted according to scientific ethics in producing and presenting the findings, cited all the references I used, express all institutions or organizations or persons who supported the thesis financially. I have never used the data and information I provide here in order to get a degree in any way.

27 May 2024

Azeez Majeed MOHAMMED



PREFACE

I would like to extend my deepest gratitude and appreciation to all the people who, in one way or another, have helped me a lot in making this research project possible.

First and foremost to God Almighty, for His unmeasurable guidance and mercy, for giving the brilliance of ideas, and for answering all my prayers. He who made all things possible;

My profound and sincere thanks to my advisers, Prof. Dr. Nevin Celik and Ass. Prof. Dr. Thamir Ahmmed, this work would not have been possible without their generous assistance in correcting this thesis, for giving valuable suggestion and encouraging comments, and for giving proper guidance that helped me go through this study despite their hectic schedule. I owe you a lot big time!

My gratitude also to Mr. Anjam Byze for his valuable suggestions and recommendations in organizing pertinent ideas in the completion of this study, my special thanks to his company members “Halgurd Company” and Mr. Ahmed Teejo”who were always give their technical support in the laboratory;

I also would like to express my sincere special thanks to Mr. Omer Azad and Mr. Abdullah Hassan members of TIU, petroleum Lab. for their kindness and warm accommodation for extending their time and efforts by accompanying us throughout NON-NEWTONIAN fluid preparation.

To my dearest Rebwar Ibrahim for his unfailing support and encouragement. His continuous guidance and support were vital.

To my siblings who served as my inspiration. For keeping me inspired to push myself to the limits and make the best of everything.

To everyone who becomes a part of my success. To my friends and to everyone who has shown support.

Most of all, I am grateful to acknowledge my beloved parents, for their love and care throughout this study. I am always grateful for the endless love invested in me that made me who I am today.

To my beautiful daughter, you are the best gif I have ever aceived in the world, your presence in my life is the reason I stand out stronger everyday, you are more than just a daughter, you are my other half, my joy and happiness. I want you to know that as far as the sky is from the earth, that is how much you mean to me, my love for you is infinity and I will always, protect you with my life, I will do anythig to see that your beautiful smile never fads away. I love you my sweetheart.you.

Azeez Majeed MOHAMMED

ELAZIG, 2024

TABLE OF CONTENTS

	Page
PREFACE	iv
TABLE OF CONTENTS	v
ABSTRACT	vii
ÖZET	viii
LIST OF FIGURES	ix
LIST OF TABLES	xvi
LIST OF APPENDICES	xvii
1. INTRODUCTION.....	1
1.1. Characteristics of Open Channel Flows.....	1
1.1.1. Quantification of open channel flows.....	2
1.1.2. Classification of open channel flows.....	5
1.2. Literature Survey	7
1.2.1. Effects of obstacles on open channel flow	7
1.2.2. Effects of fluid viscosity in open channel flows (Newtonian or non-Newtonian fluids).....	11
1.2.3. Numerical studies related to open channel flow.....	13
1.3. Aim of the Thesis	15
1.4. Objectives of the Thesis	15
2. MATERIALS AND METHODS	16
2.1. Experimental Analysis.....	18
2.1.1. Setup.....	18
2.1.2. Obstacles and their arrangements	20
2.1.3. Fluids selection and preparations	23
2.1.4. Density measurements.....	26
2.1.5. Viscosity measurements	28
2.1.6. Procedure and data reduction	30
2.2. Numerical Analysis	34
2.2.1. Physical model	34
2.2.2. Governing equations	35
2.2.3. Turbulence model.....	36
2.2.4. Two-phase flow modelling.....	37
2.2.5. Discretization	38
2.2.6. Boundary conditions	40
2.2.7. Meshing process for the selected model.....	41
2.2.8. Verification of the model	43
3. RESULTS FOR NEWTONIAN FLUIDS (WATER)	44
3.1. Velocity Variations.....	44
3.1.1. Velocity Variations along Channel Distance with Rectangle Obstacles	44
3.1.2. Velocity Variations along Channel Distance with Rectangular Arc Obstacles	59
3.1.3. Velocity Variations along Channel Distance with Cube Obstacles.....	69
3.1.4. Velocity Variations along Channel Distance with Circular Obstacles (Experimental Only).....	79

3.1.5. Velocity Variations along Channel Distance with Elliptic Obstacles (Experimental Only).....	81
3.2. Comparison of the Slope on the Velocity	84
3.3. Depth Analysis	89
3.3.1. Depth Analysis along Channel Distance with Rectangle Obstacles	89
3.3.2. Depth Analysis along Channel Distance with Rectangular Arc Obstacle	96
3.3.3. Depth Analysis along Channel Distance with Cube Obstacles	106
3.3.4. Depth Analysis along Channel Distance with Elliptic Obstacles (Experimental Only)	116
3.3.5. Depth Analysis along Channel Distance with Cylindrical Obstacles (Experimental Only).....	119
3.4. Froude (Fr) Numbers of Newtonian Fluid Flow	120
3.4.1. Importance of Froude Numbers in Open Channel Flows	120
3.4.2. Fr numbers with Rectangle Obstacle	121
3.4.3. Fr Numbers with Rectangular Arc Obstacles	126
3.4.4. Fr Numbers with Cube Obstacles	128
3.5. Comparison of All Obstacles by Means of Velocity Variations	129
4. RESULTS FOR NON-NEWTONIAN FLUIDS	134
4.1. Comparisons of Non-Newtonian Bentonite Mixtures to Water by Means of Velocity	134
4.2. Velocity Analysis	137
4.2.1. Velocity Variations along Channel Distance with Rectangular Obstacles	137
4.2.2. Velocity Variations along Channel Distance with Rectangular Arc Obstacles	144
4.2.3. Velocity Variations along Channel Distance with Cube Obstacles.....	151
4.2.4. Velocity Variations along Channel Distance with Cylindrical Obstacles (Experimental only).....	157
4.2.5. Velocity Variations along Channel Distance with Elliptical Obstacles (Experimental only).....	161
4.3. Depth Analysis of Non-Newtonian Fluids.....	162
4.3.1. Depth Analysis with Rectangular Obstacles	163
4.3.2. Depth Analysis with Rectangular Arc Obstacles.....	173
4.3.3. Depth Analysis of the Cube Obstacles	186
4.3.4. Depth Analysis of the Cylindrical Obstacles (Experimental only).....	193
4.3.5. Depth Analysis with Elliptical Obstacles (Experimental only)	195
4.4. Comparison of All Obstacles by Means of Depth Analysis	197
4.5. Comparison to the Literature	200
5. CONCLUSIONS	203
RECOMMENDATIONS	205
REFERENCES	206
APPENDICES	213
CURRICULUM VITAE	

ABSTRACT

Experimental and Numerical Investigation of Open Channel Flow Characteristics of Newtonian and Non-Newtonian Fluids

Azeez Majeed MOHAMMED

Doctoral Thesis

FIRAT UNIVERSITY

Graduate School of Natural and Applied Sciences

Department of Mechanical Engineering

May 2024, Page: xvii + 212

This thesis explores the effects of different obstacles on Newtonian and non-Newtonian fluid flow dynamics within an open channel. The experiment involved placing obstacles, rectangular shapes, rectangular arches, cubes, cylindrical, and rectangular shapes in an open channel (4 m long, 0.3 m high, and 0.2 m wide). Flow rates ranging from 0.0035 m³/s to 0.0045 m³/s were tested across slopes of 0°, 1.0156° ve 1.406°. The numerical approach complemented the experiments to ensure accurate determination of the input parameters and boundary conditions.

This thesis is organized into 5 Chapters, with each of the chapters describing a sequence of study as described as follows. Chapter 1 introduces the thesis, providing a brief overview of open channel flow and a comprehensive review of the relevant literature. Chapter 2 gives information about the experimental setup and numerical model. Chapter 3 presents the results of Newtonian fluid (water), by comparing the experimental and numerical results of velocity, water depth and Froude numbers. Chapter 4 presents the experimental and numerical analysis of the non-Newtonian fluids. Chapter 5 concludes the results.

The results of the thesis reveal several important conclusions. For example, increasing channel slopes and flow rates amplifies the effects of obstacles on velocity profiles due to increased flow separation and density around the obstacles. Near obstacles, steeper velocity gradients indicate acceleration and deceleration of flow patterns, leading to local turbulence and eddies. The presence of space between obstacles enhances shear stress along the channel bottom and walls, affecting sediment transport and erosion processes. At 0° slope, fluid velocity remains high over obstacles. With slopes greater than 0°, gravity's control over obstacle effects, especially at high flow rates, results in lower velocities after the obstacle, and sometimes lower than velocities before the obstacle. Cubic obstacle arrangements produce large velocity changes and higher fluid velocities prior the obstacles, reducing the effect of boundary layers at the channel bottom. Velocity changes in the sidewalls are affected by flow rates and obstacle arrangements due to vortex density and pressure gradients. Velocities near the wall decrease with slopes greater than 0° due to increased gravitational force.

Non-Newtonian fluids show lower velocity and depth anisotropy compared to water, which is due to higher shear stress and smoother velocity changes. Obstacles induce acceleration of the flow, resulting in higher central region velocities and lower velocities near the edges of the obstructions. Depth variations in non-Newtonian fluids are less pronounced and tend to decrease. Its depth is more sensitive to high flow rates and shows a steeper decrease with increasing gradient compared to water. This comprehensive analysis provides valuable insights into the complex dynamics of fluid flow in open channels, with major implications for engineering and environmental applications.

Keywords: Open Channel Flow, Non-Newtonian Fluids, Obstacle Effect, Slope Effect

ÖZET

Newtonsal ve Newtonsal Olmayan Akışkanların Açık Kanal Akış Karakteristiklerinin Deneysel ve Sayısal İncelenmesi

Azeez Majeed MOHAMMED

Doktora Tezi

FIRAT ÜNİVERSİTESİ

Fen Bilimleri Enstitüsü

Makina Mühendisliği Anabilim Dalı

Mayıs 2024, Sayfa: xvii + 212

Bu tez, açık bir kanal içerisinde farklı engellerin Newton tipi ve Newton tipi olmayan akışkan akış dinamiği üzerindeki etkilerini araştırmaktadır. Deney, açık bir kanala (4 m uzunluğunda, 0.3 m yüksekliğinde ve 0.2 m genişliğinde) engellerin, dikdörtgen şekillerin, dikdörtgen kemerlerin, küplerin, silindirik ve dikdörtgen şekillerin yerleştirilmesini içermektedir. $0.0035 \text{ m}^3/\text{s}$ ile $0.0045 \text{ m}^3/\text{s}$ arasında değişen akış debileri 0° , 1.0156° ve 1.406° eğimlerde test edilmiştir. Sayısal yaklaşım, giriş parametrelerinin ve sınır koşullarının doğru şekilde belirlenmesini sağlamak için deneyleri tamamlamıştır.

Bu tez 5 Bölüm halinde düzenlenmiştir ve bölümlerin her biri aşağıda açıklandığı gibi bir çalışma dizisini açıklamaktadır. Bölüm 1, açık kanal akışına kısa bir genel bakış ve ilgili literatürün kapsamlı bir incelemesini sunarak tezi tanıtmaktadır. Bölüm 2'de deney düzeneği ve sayısal model hakkında bilgi verilmektedir. Bölüm 3, hız, su derinliği ve Froude sayılarının deneysel ve sayısal sonuçlarını karşılaştırarak Newton tipi akışkanın (su) sonuçlarını sunmaktadır. Bölüm 4'te Newtonsal olmayan akışkanların deneysel ve sayısal analizleri sunulmaktadır. Bölüm 5'te tüm sonuçlar değerlendirilmektedir..

Tezin sonuçları birçok önemli değerlendirmeyi ortaya koymaktadır. Örneğin, artan kanal eğimi ve debi, engel etrafında artan akış ayrılmaları nedeniyle engellerin hız profilleri üzerindeki etkisini artırır. Engellerin yakınında, daha dik hız gradyanları akış düzenlerinin hızlandığını ve yavaşladığını gösterir, bu da yerel türbülansa ve girdaplara yol açar. Engeller arasındaki boşluğun varlığı, taban ve duvarlar boyunca kayma gerilimini artırarak tortu taşınmasını ve erozyon süreçlerini etkiler. 0° eğimde, engellerin üzerinden geçerken sıvı hızı yüksek kalır. 0° 'den büyük eğimlerde, özellikle yüksek debilerde, yer çekiminin engel üzerindeki kontrolü, engelden sonraki hızların ve bazen engel önündeki hızların daha düşük olmasına neden olur. Kübik engel düzenlemeleri, engellerden önce büyük hız değişiklikleri üreterek kanal tabanındaki sınır tabakanın etkisini azaltır. Artan yerçekimi nedeniyle 0° 'den büyük eğimlerde duvara yakın hızlar azalır.

Newtonsal olmayan akışkanlar, suya kıyasla daha düşük hız ve derinlik anizotropisi gösterir; bunun nedeni, daha yüksek kayma gerilimi ve daha yumuşak hız değişimleridir. Engeller akışın hızlanmasına neden olur, bu da merkezi bölge hızlarının daha yüksek olmasına ve engellerin kenarlarına yakın yerlerde daha düşük hızlara neden olur. Newtonsal olmayan akışkanlarda derinlik değişimleri daha az belirgindir ve azalma eğilimindedir. Derinliği yüksek akış hızlarına daha duyarlıdır ve suya göre artan eğimle daha dik bir azalma gösterir. Bu kapsamlı analiz, açık kanallardaki sıvı akışının karmaşık dinamikleri hakkında, mühendislik ve çevre uygulamaları için önemli sonuçlar doğuracak değerli bilgiler sağlar.

Anahtar Kelimeler: Açık kanal akışı, Newtonsal olmayan akışkan, Engelli Etkisi, Eğim Etkisi

LIST OF FIGURES

	Page
Figure 1.1. Open channel flow sketch.....	1
Figure 1.2. Profile view of quantities in basic open channel flow.....	3
Figure 1.3. View of basic open channel flow quantities cross-section	3
Figure 2.1. Experimental setup of the open channel in the laboratory	19
Figure 2.2. Schematic view of the open channel system	19
Figure 2.3. Layout of obstacles; a) cube, b) short cylindrical, c) arc plate, d) rectangular plate, ...	22
Figure 2.4. Arrangement of measurements area	23
Figure 2.5. Viscometer used for viscosity determination.	24
Figure 2.6. Viscometer with a) long cylindrical spindle, b) spindle with four perpendicular plates, c) very short cylindrical spindle	25
Figure 2.7. Density determination apparatus	27
Figure 2.8. Specific gravity of bentonite mixtures	27
Figure 2.9. The density measurement cup	28
Figure 2.10. Variation of kinematic viscosity for different viscometer spindles	29
Figure 2.11. Digital current meter	30
Figure 2.12. Digital velocity meter, flow depth point gauge, and magnetic flowmeter for water measurements.	31
Figure 2.13. Sample of Excel file used for measurement documentation.	33
Figure 2.14. Solution domain a) from the CFX, b) schematic view with boundary conditions	40
Figure 2.15. Mesh structure views from the numerical analysis	42
Figure 3.1. Variation of velocity for rectangular obstacles at $Q = 0.0035 \text{ m}^3/\text{s}$, $S = 0^\circ$	45
Figure 3.2. Variation of velocity for rectangular obstacles at $Q = 0.004 \text{ m}^3/\text{s}$, $S = 0^\circ$	46
Figure 3.3. Variation of velocity for rectangular obstacles at $Q = 0.0045 \text{ m}^3/\text{s}$, $S = 0^\circ$	47
Figure 3.4. Variation of velocity for rectangular obstacles at $Q = 0.0035 \text{ m}^3/\text{s}$, $S = 1.0156^\circ$	48
Figure 3.5. Variation of velocity for rectangular obstacles at $Q = 0.004 \text{ m}^3/\text{s}$, $S = 1.0156^\circ$	49
Figure 3.6. Variation of velocity for rectangular obstacles at $Q = 0.0045 \text{ m}^3/\text{s}$, $S = 1.0156^\circ$	49
Figure 3.7. Variation of velocity for rectangular obstacles at $Q = 0.0035 \text{ m}^3/\text{s}$, $S = 1.406^\circ$	50
Figure 3.8. Variation of velocity for rectangular obstacles at $Q = 0.004 \text{ m}^3/\text{s}$, $S = 1.406^\circ$	51
Figure 3.9. Variation of velocity for rectangular obstacles at $Q = 0.0045 \text{ m}^3/\text{s}$, $S = 1.406^\circ$	52
Figure 3.10. Experimental variation of stream lines for rectangular obstacle $Q = 0.004 \text{ m}^3/\text{s}$	54
Figure 3.11. Numerical variation of velocity contours for rectangular obstacle $Q = 0.0035 \text{ m}^3/\text{s}$	55

Figure 3.12. Numerical variation of stream lines for rectangular obstacle $Q = 0.0035 \text{ m}^3/\text{s}$	56
Figure 3.13. Numerical variation of velocity contours for rectangular obstacle, $Q = 0.004 \text{ m}^3/\text{s}$	57
Figure 3.14. Numerical variation of stream lines for rectangular obstacle, $Q = 0.004 \text{ m}^3/\text{s}$	57
Figure 3.15. Numerical variation of velocity contours for rectangular obstacle, $Q = 0.0045 \text{ m}^3/\text{s}$	58
Figure 3.16. Numerical variation of stream lines for rectangular obstacle, $Q = 0.0045 \text{ m}^3/\text{s}$	58
Figure 3.17. Variation of velocity for rectangular arc obstacles at $Q = 0.0035 \text{ m}^3/\text{s}$, $S = 0^\circ$	59
Figure 3.18. Variation of velocity for rectangular arc obstacles at $Q = 0.004 \text{ m}^3/\text{s}$, $S = 0^\circ$	60
Figure 3.19. Variation of velocity for rectangular arc obstacles at $Q = 0.0045 \text{ m}^3/\text{s}$, $S = 0^\circ$	61
Figure 3.20. Variation of velocity for rectangular arc obstacles at $Q = 0.0035 \text{ m}^3/\text{s}$, $S = 1.0156^\circ$	62
Figure 3.21. Variation of velocity for rectangular arc obstacles at $Q = 0.004 \text{ m}^3/\text{s}$, $S = 1.0156^\circ$	62
Figure 3.22. Variation of velocity for rectangular arc obstacles at $Q = 0.0045 \text{ m}^3/\text{s}$, $S = 1.0156^\circ$	63
Figure 3.23. Variation of velocity for rectangular arc obstacles at $Q = 0.0035 \text{ m}^3/\text{s}$, $S = 1.406^\circ$	64
Figure 3.24. Variation of velocity for rectangular arc obstacles at $Q = 0.004 \text{ m}^3/\text{s}$, $S = 1.406^\circ$	64
Figure 3.25. Variation of velocity for rectangular arc obstacles at $Q = 0.0045 \text{ m}^3/\text{s}$, $S = 1.406^\circ$	65
Figure 3.26. Numerical variation of velocity contours for rectangular arc obstacle $Q = 0.0035 \text{ m}^3/\text{s}$	66
Figure 3.27. Numerical variation of stream lines for rectangular arc obstacle $Q = 0.0035 \text{ m}^3/\text{s}$	66
Figure 3.28. Numerical variation of velocity contours for rectangular arc obstacle $Q = 0.004 \text{ m}^3/\text{s}$	67
Figure 3.29. Numerical variation of stream lines for rectangular arc obstacle $Q = 0.004 \text{ m}^3/\text{s}$	67
Figure 3.30. Numerical variation of velocity contours for rectangular arc obstacle $Q = 0.0045 \text{ m}^3/\text{s}$	68
Figure 3.31. Numerical variation of stream lines for rectangular arc obstacle $Q = 0.0045 \text{ m}^3/\text{s}$	68
Figure 3.32. Variation of velocity for cube obstacles at $Q = 0.0035 \text{ m}^3/\text{s}$, $S = 0^\circ$	70
Figure 3.33. Variation of velocity for cube obstacles at $Q = 0.004 \text{ m}^3/\text{s}$, $S = 0^\circ$	70
Figure 3.34. Variation of velocity for cube obstacles at $Q = 0.0045 \text{ m}^3/\text{s}$, $S = 0^\circ$	71
Figure 3.35. Variation of velocity for cube obstacles at $Q = 0.0035 \text{ m}^3/\text{s}$, $S = 1.0156^\circ$	71
Figure 3.36. Variation of velocity for cube obstacles at $Q = 0.004 \text{ m}^3/\text{s}$, $S = 1.0156^\circ$	72
Figure 3.37. Variation of velocity for cube obstacles at $Q = 0.0045 \text{ m}^3/\text{s}$, $S = 1.0156^\circ$	73
Figure 3.38. Variation of velocity for cube obstacles at $Q = 0.0035 \text{ m}^3/\text{s}$, $S = 1.406^\circ$	73
Figure 3.39. Variation of velocity for cube obstacles at $Q = 0.004 \text{ m}^3/\text{s}$, $S = 1.406^\circ$	74
Figure 3.40. Variation of velocity for cube obstacles at $Q = 0.0045 \text{ m}^3/\text{s}$, $S = 1.406^\circ$	75
Figure 3.41. Numerical variation of velocity contours for cubic obstacle $Q = 0.0035 \text{ m}^3/\text{s}$	76
Figure 3.42. Numerical variation of stream lines for cubic obstacle $Q = 0.0035 \text{ m}^3/\text{s}$	76
Figure 3.43. Numerical variation of velocity contours for cubic obstacle $Q = 0.004 \text{ m}^3/\text{s}$	77
Figure 3.44. Numerical variation of stream lines for cubic obstacle $Q = 0.004 \text{ m}^3/\text{s}$	77

Figure 3.45. Numerical variation of velocity contours for cubic obstacle $Q = 0.0045 \text{ m}^3/\text{s}$	78
Figure 3.46. Numerical variation of stream lines for cubic obstacle $Q = 0.0045 \text{ m}^3/\text{s}$	78
Figure 3.47. Variation of velocity for cylindrical obstacle $Q = 0.0035 \text{ m}^3/\text{s}$	79
Figure 3.48. Variation of velocity for cylindrical obstacle $Q = 0.004 \text{ m}^3/\text{s}$	80
Figure 3.49. Variation of velocity for cylindrical obstacle $Q = 0.0045 \text{ m}^3/\text{s}$	81
Figure 3.50. Variation of water flow velocity for elliptical obstacle $Q = 0.0035 \text{ m}^3/\text{s}$	82
Figure 3.51. Variation of water flow velocity, Elliptical obstacle, $Q = 0.004 \text{ m}^3/\text{s}$	83
Figure 3.52. Variation of water flow velocity ,Elliptical obstacle, $Q = 0.0045 \text{ m}^3/\text{s}$	84
Figure 3.53. Top view flow at a) $S = 1.406^\circ$ with $Q = 0.0035 \text{ m}^3/\text{s}$, b) $Q = 0.0045 \text{ m}^3/\text{s}$	85
Figure 3.54. Water profiles from top view $S = 1.406^\circ$, $Q = 0.0045 \text{ m}^3/\text{s}$	85
Figure 3.55. Depth profile from side view for $Q = 0.0035 \text{ m}^3/\text{s}$, a) $S = 0^\circ$, b) $S = 1.0156^\circ$, c) $S = 1.406^\circ$	86
Figure 3.56. Depth profile in first obstacle explaining the flood jump, a) $S = 1.0156^\circ$, $Q = 0.0035 \text{ m}^3/\text{s}$	86
Figure 3.57. Water side profile at first obstacle, $S = 1.406^\circ$, $Q = 0.004 \text{ m}^3/\text{s}$	87
Figure 3.58. Water side profile at first obstacle, $S = 1.406^\circ$, $Q = 0.0045 \text{ m}^3/\text{s}$	87
Figure 3.59. Velocity profile from top view for $Q = 0.0035 \text{ m}^3/\text{s}$, a) $S = 0^\circ$, b) $S = 1.0156^\circ$, c) $S = 1.406^\circ$	88
Figure 3.60. Velocity profile from top view for $Q = 0.004 \text{ m}^3/\text{s}$, a) $S = 0^\circ$, b) $S = 1.0156^\circ$, c) $S = 1.406^\circ$	88
Figure 3.61. Velocity profile from top view for $Q = 0.0045 \text{ m}^3/\text{s}$, a) $S = 0^\circ$, b) $S = 1.0156^\circ$, c) $S = 1.406^\circ$	88
Figure 3.62. Water depth along channel for rectangular obstacles; $Q = 0.0035 \text{ m}^3/\text{s}$, $S = 0^\circ$	89
Figure 3.63. Water depth along channel for rectangular obstacles; $Q = 0.004 \text{ m}^3/\text{s}$, $S = 0^\circ$	90
Figure 3.64. Water depth along channel for rectangular obstacles; $Q = 0.0045 \text{ m}^3/\text{s}$, $S = 0^\circ$	91
Figure 3.65. Water depth along channel for rectangular obstacles; $Q = 0.0035 \text{ m}^3/\text{s}$, $S = 1.0156^\circ$	92
Figure 3.66. Water depth along channel for rectangular obstacles; $Q = 0.004 \text{ m}^3/\text{s}$, $S = 1.0156^\circ$	93
Figure 3.67. Water depth along channel for rectangular obstacles; $Q = 0.0045 \text{ m}^3/\text{s}$, $S = 1.0156^\circ$	93
Figure 3.68. Water depth along channel for rectangular obstacles; $Q = 0.0035 \text{ m}^3/\text{s}$, $S = 1.406^\circ$	94
Figure 3.69. Water depth along channel for rectangular obstacles; $Q = 0.004 \text{ m}^3/\text{s}$, $S = 1.406^\circ$	95
Figure 3.70. Water depth along channel for rectangular obstacles; $Q = 0.0045 \text{ m}^3/\text{s}$, $S = 1.406^\circ$	95
Figure 3.71. Numerically comparison of all depth values a) $S = 0^\circ$, b) $S = 1.0156^\circ$, c) $S = 1.406^\circ$	96
Figure 3.72. Water depth along channel for rectangular arc obstacles; $Q = 0.0035 \text{ m}^3/\text{s}$, $S = 0^\circ$	97
Figure 3.73. Water depth along channel for rectangular arc obstacles; $Q = 0.004 \text{ m}^3/\text{s}$, $S = 0^\circ$	98

Figure 3.74. Water depth along channel for rectangular arc obstacles; $Q = 0.0045 \text{ m}^3/\text{s}$, $S = 0^\circ$	99
Figure 3.75. Water depth along channel for rectangular arc obstacles; $Q = 0.0035 \text{ m}^3/\text{s}$, $S = 1.0156^\circ$...	100
Figure 3.76. Water depth along channel for rectangular arc obstacles; $Q = 0.004 \text{ m}^3/\text{s}$, $S = 1.0156^\circ$	101
Figure 3.77. Water depth along channel for rectangular arc obstacles; $Q = 0.0045 \text{ m}^3/\text{s}$, $S = 1.0156^\circ$...	102
Figure 3.78. Water depth along channel for rectangular arc obstacles; $Q = 0.0035 \text{ m}^3/\text{s}$, $S = 1.406^\circ$	103
Figure 3.79. Water depth along channel for rectangular arc obstacles; $Q = 0.004 \text{ m}^3/\text{s}$, $S = 1.406^\circ$	103
Figure 3.80. Water depth along channel for rectangular arc obstacles; $Q = 0.0045 \text{ m}^3/\text{s}$, $S = 1.406^\circ$	104
Figure 3.81. Depth for the rectangular arc obstacle, from the side view for $Q = 0.0035 \text{ m}^3/\text{s}$	104
Figure 3.82. Depth for the rectangular arc obstacle, from the side view for $Q = 0.004 \text{ m}^3/\text{s}$	105
Figure 3.83. Depth for the rectangular arc obstacle, from the side view for $Q = 0.0045 \text{ m}^3/\text{s}$	105
Figure 3.84. Water depth along channel for cubic obstacles; $Q = 0.0035 \text{ m}^3/\text{s}$, $S = 0^\circ$	106
Figure 3.85. Water depth along channel for cubic obstacles; $Q = 0.004 \text{ m}^3/\text{s}$, $S = 0^\circ$	107
Figure 3.86. Water depth along channel for cubic obstacles; $Q = 0.0045 \text{ m}^3/\text{s}$, $S = 0^\circ$	108
Figure 3.87. Water depth along channel for cubic obstacles; $Q = 0.0035 \text{ m}^3/\text{s}$, $S = 1.0156^\circ$	109
Figure 3.88. Water depth along channel for cubic obstacles; $Q = 0.004 \text{ m}^3/\text{s}$, $S = 1.0156^\circ$	110
Figure 3.89. Water depth along channel for cubic obstacles; $Q = 0.0045 \text{ m}^3/\text{s}$, $S = 1.0156^\circ$	111
Figure 3.90. Water depth along channel for cubic obstacles; $Q = 0.0035 \text{ m}^3/\text{s}$, $S = 1.406^\circ$	112
Figure 3.91. Water depth along channel for cubic obstacles; $Q = 0.004 \text{ m}^3/\text{s}$, $S = 1.406^\circ$	113
Figure 3.92. Water depth along channel for cubic obstacles; $Q = 0.0045 \text{ m}^3/\text{s}$, $S = 1.406^\circ$	114
Figure 3.93. Depth for the rectangular arc obstacle, from the side view for; a) $Q = 0.0035 \text{ m}^3/\text{s}$; b) $Q = 0.004 \text{ m}^3/\text{s}$, c) $Q = 0.0045 \text{ m}^3/\text{s}$	116
Figure 3.94. Variation of water flow depth for elliptical obstacle $Q = 0.0035 \text{ m}^3/\text{s}$	117
Figure 3.95. Variation of water flow depth for elliptical obstacle $Q = 0.004 \text{ m}^3/\text{s}$	118
Figure 3.96. Variation of water flow depth for elliptical obstacle $Q = 0.004 \text{ m}^3/\text{s}$	119
Figure 3.97. Velocity and depth values for cylindrical obstacle.....	119
Figure 3.98. The rectangle obstacle with all flowrates and slopes	130
Figure 3.99. The rectangular arc obstacle with all flowrates and slopes	130
Figure 3.100. The cubic obstacle with all flowrates and slopes.....	131
Figure 3.101. Flow after the rectangle arc obstacles, a) Swirl formation, b) Bubbles inside the water c) Separations.....	132
Figure 3.102. Vortices profile in the arc plate obstacle at different flows and depth for flow at zone 7.5 cm far from the wall.	133
Figure 4.1. Velocities for all considered liquids, (rectangular arc obstacle, $Q = 0.0035 \text{ m}^3/\text{s}$, $S = 0^\circ$)..	134

Figure 4.2.	Velocities for all considered liquids, (rectangular arc obstacle, $Q = 0.0035 \text{ m}^3/\text{s}$, $S = 1.0156^\circ$)	135
Figure 4.3.	Velocities for all considered liquids, (rectangular arc obstacle, $Q = 0.0035 \text{ m}^3/\text{s}$, $S = 1.406^\circ$)	135
Figure 4.4.	Velocities for all considered liquids, (rectangular arc obstacle, $Q = 0.004 \text{ m}^3/\text{s}$, $S = 1.406^\circ$)	136
Figure 4.5.	Velocities for all considered liquids, (rectangular arc obstacle, $Q = 0.0045 \text{ m}^3/\text{s}$, $S = 1.406^\circ$)	136
Figure 4.6.	Variation of NN velocity for rectangular obstacles at $Q = 0.0035 \text{ m}^3/\text{s}$, $S = 0^\circ$	137
Figure 4.7.	Variation of NN velocity for rectangular obstacles at $Q = 0.004 \text{ m}^3/\text{s}$, $S = 0^\circ$	139
Figure 4.8.	Variation of NN velocity for rectangular obstacles at $Q = 0.0045 \text{ m}^3/\text{s}$, $S = 0^\circ$	139
Figure 4.9.	Variation of NN velocity for rectangular obstacles at $Q = 0.0035 \text{ m}^3/\text{s}$, $S = 1.0156^\circ$	140
Figure 4.10.	Variation of NN velocity for rectangular obstacles at $Q = 0.004 \text{ m}^3/\text{s}$, $S = 1.0156^\circ$	141
Figure 4.11.	Variation of NN velocity for rectangular obstacles at $Q = 0.0045 \text{ m}^3/\text{s}$, $S = 1.0156^\circ$	142
Figure 4.12.	Variation of NN velocity for rectangular obstacles at $Q = 0.0035 \text{ m}^3/\text{s}$, $S = 1.406^\circ$	143
Figure 4.13.	Variation of NN velocity for rectangular obstacles at $Q = 0.004 \text{ m}^3/\text{s}$, $S = 1.406^\circ$	143
Figure 4.14.	Variation of NN velocity for rectangular obstacles at $Q = 0.0045 \text{ m}^3/\text{s}$, $S = 1.406^\circ$	144
Figure 4.15.	Variation of NN velocity for rectangular arc obstacles at $Q = 0.0035 \text{ m}^3/\text{s}$, $S = 0^\circ$	145
Figure 4.16.	Variation of NN velocity for rectangular arc obstacles at $Q = 0.004 \text{ m}^3/\text{s}$, $S = 0^\circ$	145
Figure 4.17.	Variation of NN velocity for rectangular arc obstacles at $Q = 0.0045 \text{ m}^3/\text{s}$, $S = 0^\circ$	146
Figure 4.18.	Variation of NN velocity for rectangular arc obstacles at $Q = 0.0035 \text{ m}^3/\text{s}$, $S = 1.0156^\circ$	146
Figure 4.19.	Variation of NN velocity for rectangular arc obstacles at $Q = 0.004 \text{ m}^3/\text{s}$, $S = 1.0156^\circ$	147
Figure 4.20.	Variation of NN velocity for rectangular arc obstacles at $Q = 0.0045 \text{ m}^3/\text{s}$, $S = 1.0156^\circ$	147
Figure 4.21.	Variation of NN velocity for rectangular arc obstacles at $Q = 0.0035 \text{ m}^3/\text{s}$, $S = 1.406^\circ$	148
Figure 4.22.	Variation of NN velocity for rectangular arc obstacles at $Q = 0.004 \text{ m}^3/\text{s}$, $S = 1.406^\circ$	148
Figure 4.23.	Variation of NN velocity for rectangular arc obstacles at $Q = 0.0045 \text{ m}^3/\text{s}$, $S = 1.406^\circ$	149
Figure 4.24.	Velocity profile at slope $S = 0^\circ$, $Q = 0.0035 \text{ m}^3/\text{s}$ in a) center, b) side wall, and c) top view	149
Figure 4.25.	Velocity profile at slope $S = 1.0156^\circ$, $Q = 0.0035 \text{ m}^3/\text{s}$ in a) center, b) side wall, and c) top view	150
Figure 4.26.	Velocity profile at slope $S = 1.406^\circ$, $Q = 0.0035 \text{ m}^3/\text{s}$ in a) center, b) side wall, and c) top view	150
Figure 4.27.	Vortex profiles at slope a) $S = 0^\circ$ b) $S = 1.0156^\circ$ c) $S = 1.406^\circ$	151
Figure 4.28.	Variation of NN velocity for cube obstacles at $Q = 0.0035 \text{ m}^3/\text{s}$, $S = 0^\circ$	152

Figure 4.29. Variation of NN velocity for cube obstacles at $Q = 0.004 \text{ m}^3/\text{s}$, $S = 0^\circ$	152
Figure 4.30. Variation of NN velocity for cube obstacles at $Q = 0.0045 \text{ m}^3/\text{s}$, $S = 0^\circ$	153
Figure 4.31. Variation of NN velocity for cube obstacles at $Q = 0.0035 \text{ m}^3/\text{s}$, $S = 1.0156^\circ$	153
Figure 4.32. Variation of NN velocity for cube obstacles at $Q = 0.004 \text{ m}^3/\text{s}$, $S = 1.0156^\circ$	154
Figure 4.33. Variation of NN velocity for cube obstacles at $Q = 0.0045 \text{ m}^3/\text{s}$, $S = 1.0156^\circ$	155
Figure 4.34. Variation of NN velocity for cube obstacles at $Q = 0.0035 \text{ m}^3/\text{s}$, $S = 1.406^\circ$	156
Figure 4.35. Variation of NN velocity for Cube obstacles at $Q = 0.004 \text{ m}^3/\text{s}$, $S = 1.046^\circ$	157
Figure 4.36. Variation of NN velocity for cube obstacles at $Q = 0.0045 \text{ m}^3/\text{s}$, $S = 1.046^\circ$	157
Figure 4.37. Variation of Newtonian and Non Newtonian flow velocity for cylindrical obstacle with $Q = 0.0035 \text{ m}^3/\text{s}$	158
Figure 4.38. Variation of Newtonian and Non Newtonian flow velocity for cylindrical obstacle with $Q = 0.004 \text{ m}^3/\text{s}$	159
Figure 4.39. Variation of Newtonian and Non Newtonian flow velocity for cylindrical obstacle with $Q = 0.0045 \text{ m}^3/\text{s}$	160
Figure 4.40. Variation of Newtonian and Non Newtonian flow velocity for elliptical obstacle with $Q = 0.0035 \text{ m}^3/\text{s}$	161
Figure 4.41. Variation of Newtonian and Non Newtonian flow velocity for elliptical obstacle with $Q = 0.004 \text{ m}^3/\text{s}$	162
Figure 4.42. Variation of Newtonian and Non Newtonian flow velocity for elliptical obstacle with $Q = 0.0045 \text{ m}^3/\text{s}$	162
Figure 4.43. Variation of NN depth for rectangular obstacles at $Q = 0.0035 \text{ m}^3/\text{s}$, $S = 0^\circ$	164
Figure 4.44. Variation of NN depth velocity for rectangular obstacles at $Q = 0.004 \text{ m}^3/\text{s}$, $S = 0^\circ$	165
Figure 4.45. Variation of NN depth for rectangular obstacles at $Q = 0.0045 \text{ m}^3/\text{s}$, $S = 0^\circ$	166
Figure 4.46. Variation of NN depth for rectangular obstacles at $Q = 0.0035 \text{ m}^3/\text{s}$, $S = 1.0156^\circ$	166
Figure 4.47. Variation of NN depth for rectangular obstacles at $Q = 0.004 \text{ m}^3/\text{s}$, $S = 1.0156^\circ$	167
Figure 4.48. Variation of NN depth for rectangular obstacles at $Q = 0.0045 \text{ m}^3/\text{s}$, $S = 1.0156^\circ$	168
Figure 4.49. Variation of NN depth for rectangular obstacles at $Q = 0.0035 \text{ m}^3/\text{s}$, $S = 1.406^\circ$	169
Figure 4.50. Variation of NN depth for rectangular obstacles at $Q = 0.004 \text{ m}^3/\text{s}$, $S = 1.406^\circ$	170
Figure 4.51. Variation of NN depth for rectangular obstacles at $Q = 0.0045 \text{ m}^3/\text{s}$, $S = 1.406^\circ$	171
Figure 4.52. Variation of NN depth for rectangular arc obstacles at $Q = 0.0035 \text{ m}^3/\text{s}$, $S = 0^\circ$	174
Figure 4.53. Variation of NN depth for rectangular arc obstacles at $Q = 0.004 \text{ m}^3/\text{s}$, $S = 0^\circ$	175
Figure 4.54. Variation of NN depth for rectangular arc obstacles at $Q = 0.0045 \text{ m}^3/\text{s}$, $S = 0^\circ$	176
Figure 4.55. Variation of NN depth for rectangular arc obstacles at $Q = 0.0035 \text{ m}^3/\text{s}$, $S = 1.0156^\circ$	177
Figure 4.56. Variation of NN depth for rectangular arc obstacles at $Q = 0.004 \text{ m}^3/\text{s}$, $S = 1.0156^\circ$	179

Figure 4.57. Variation of NN depth for rectangular arc obstacles at $Q = 0.0045 \text{ m}^3/\text{s}$, $S=1.0156^\circ$	180
Figure 4.58. Variation of NN depth for rectangular arc obstacles at $Q = 0.0035 \text{ m}^3/\text{s}$, $S=1.406^\circ$	181
Figure 4.59. Variation of NN depth for rectangular arc obstacles $Q = 0.004 \text{ m}^3/\text{s}$, $S=1.406^\circ$	182
Figure 4.60. Variation of NN depth for rectangular arc obstacles at $Q = 0.0045 \text{ m}^3/\text{s}$, $S=1.406^\circ$	183
Figure 4.61. Depth profiles at slope a) $S = 0$, b) $S = 1.0156^\circ$ c) $S = 1.406^\circ$ for $Q = 0.0035 \text{ m}^3/\text{s}$	183
Figure 4.62. Depth profiles at slope a) $S = 0$, b) $S = 1.0156^\circ$ c) $S = 1.406^\circ$ for $Q = 0.0040 \text{ m}^3/\text{s}$	184
Figure 4.63. Depth profiles at slopes a) $S = 0$, b) $S = 1.0156^\circ$ c) $S = 1.406^\circ$ for $Q = 0.0045 \text{ m}^3/\text{s}$	184
Figure 4.64. Photos from lab. a) Depth measuring b) bentonite after running proses (at rest) c) bentonite starting running flow ($S = 1.406^\circ$ for $Q = 0.0045 \text{ m}^3/\text{s}$)	185
Figure 4.65. Top view profiles of bentonite in different slope at full running process with a) $S = 1.406^\circ$ b) $S = 1.0156^\circ$ c) $S = 0$, for $Q = 0.0045 \text{ m}^3/\text{s}$	185
Figure 4.66. Variation of NN depth for cube obstacles at $Q = 0.0035 \text{ m}^3/\text{s}$, $S=0^\circ$	186
Figure 4.67. Variation of NN depth for cube obstacles at $Q = 0.004 \text{ m}^3/\text{s}$, $S=0^\circ$	187
Figure 4.68. Variation of NN depth for cube obstacles at $Q = 0.0045 \text{ m}^3/\text{s}$, $S=0^\circ$	188
Figure 4.69. Variation of NN depth for cube obstacles at $Q = 0.0035 \text{ m}^3/\text{s}$, $S=1.0156^\circ$	188
Figure 4.70. Variation of NN depth for cube obstacles at $Q = 0.004 \text{ m}^3/\text{s}$, $S=1.0156^\circ$	189
Figure 4.71. Variation of NN depth for cube obstacles at $Q = 0.0045 \text{ m}^3/\text{s}$, $S=1.0156^\circ$	190
Figure 4.72. Variation of NN depth for cube obstacles at $Q = 0.0035 \text{ m}^3/\text{s}$, $S=1.406^\circ$	191
Figure 4.73. Variation of NN depth for cube obstacles at $Q = 0.004 \text{ m}^3/\text{s}$, $S=1.406^\circ$	192
Figure 4.74. Variation of NN depth for cube obstacles at $Q = 0.0045 \text{ m}^3/\text{s}$, $S=1.406^\circ$	193

LIST OF TABLES

	Page
Table 2.1. Studied cases.....	16
Table 2.2. Bentonite concentration ratios and properties	23
Table 2.3. Mixing ratios and densities of considered mixtures.....	25
Table 2.4. Kinematic viscosity for different viscometer spendles	29



LIST OF APPENDICES

	Page
Appendix- 1. Froude number for all water cases (Rectangular obstacle)	213



1. INTRODUCTION

1.1. Characteristics of Open Channel Flows

A core component of any major drainage system is open channels. Open channels are the most common major drainage system component used to transport all the storm-water runoff collected in drainage systems. Open channels are versatile and come in several different types and consist of several different channel components. Open channels are in effect the final instrument within a drainage system for handling storm-water and as such have the final interaction with storm-water before it flows into a major river or other large body of water [1], [2] While the primary function of open channels is conveyance of runoff, many design decisions contribute to the role of channels in the urban environment in terms of stability, multiple use benefits, social acceptance, aesthetics, resource management, and maintenance [3], [4].

An open channel flow is a flow system in which the top surface of a fluid exposed directly to constant pressure. The liquid flow is partially enclosed by solid boundaries (such as in canals, river, gutters, culverts etc.,). The flowing liquid has a free surface driven by gravitational force, and thus its surface is not under any gauge pressure [5][6][7].

According to Moglen [1] no pressure variation that will build up between two sections across the channel and the same resistance can be employed. To overcome the resistance and simultaneously cause flow of the liquid in a channel. The channel is constructed in such a way that its bottom inclined towards the direction of the flow. This will enable the component of weight of the flowing liquid to move in a desire direction of the flow [8], [9].

Figure 1.1 shows a sketch of an open channel flow indicating water flow, depth-free surface, and velocity distribution.

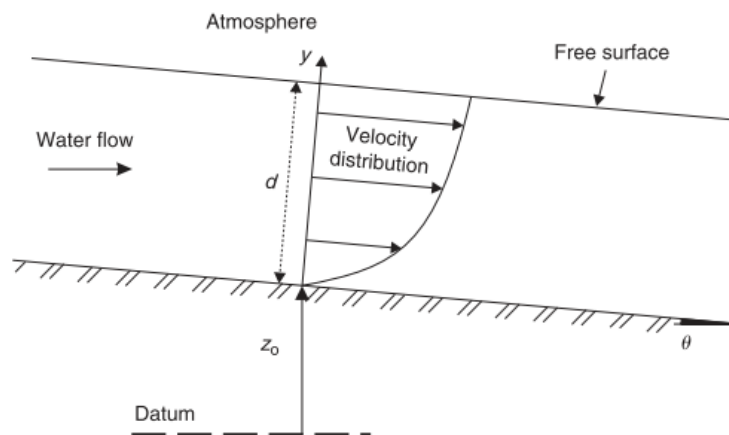


Figure 1.1. Open channel flow sketch

Open channel flows also known as free surface flow are flows in rivers, artificial channels, streams, irrigation ditches, partly filled pipes, etc. The surface of the flow experiences a uniform atmospheric pressure [10]. Turbulence is widespread in environmental flows such as in rivers, lakes, estuaries, and oceans, and it is associated with chaotic and random motions, which enhances the mixing and transport of momentum and mass [11]. The interaction of various natural and man-made obstacles encountered in the real world with the surrounding environmental flows can lead to the generation of turbulence which determines local or even global flow patterns [12].

Open channel takes place due to slope of bed of the channel only. The hydraulic grade line of open channel flow is exactly coinciding with the water surface, but the total energy lines lie at a distance of above the hydraulic grade line at every section as in closed channel. The longitudinal profile of the free surface in an open channel flow defines the hydraulic gradient and determines the cross-sectional area of flow. It necessitates the introduction of an extra variable, the stage, to define the position of the free surface at any point in the channel. Consequently, problems in open channel flow are more complex, and the solutions are more diverse which makes its study both interesting and challenging.

1.1.1. Quantification of open channel flows

The primary factor in open channel flow analysis is the location of the free surface. The free surface rises and falls in response to perturbations to the flow (variations in channel slope or width). The fundamental parameters of a hydraulic study are the geometry of the channel (slope, width, and roughness), the properties of the flowing fluid (e.g. viscosity and density) and the flow parameters (velocity and flow depth) [13]. Fig. 1.2 illustrates a longitudinal view in an open channel flow. The figure displays various basic quantities that govern the solutions to problems in open channel flow. The flow depth, y , is perhaps considered most significant quantity when dealing with open channel flow problems. Depth is measured vertically from channel bottom to the water surface. The quantity, z , represents any topography offered by the channel bottom relative to the horizontal datum [14]. The flow velocity denoted by letter v , is equally vital parameter in solving problems in open channel flow. It's obviously that velocity differs in the vertical from zero at the channel bottom to maximum value near or at the water surface. Similarly, velocity varies across channel width in very complex ways depending upon the channel shape [15], [16].

Figure 1.3 shows a cross-sectional view of flow in an open channel. The channel cross-section is quite complex in shape. The flow depth y , determines cross-sectional area of the channel, A . In contrast, a closed conduit like a pipe, presumed to be flowing full, its cross-sectional area is fixed and constant. Thus, the physics of the flow are better understood with deep

knowledge of the flow velocity, total energy possessed by the flow and issues of friction. In open channel flow, the depth plays a dual role: (i) regulates the flow energy, and (ii) it determines the flowing volume cross-sectional area. This is what makes solving problems in open channel more challenging.

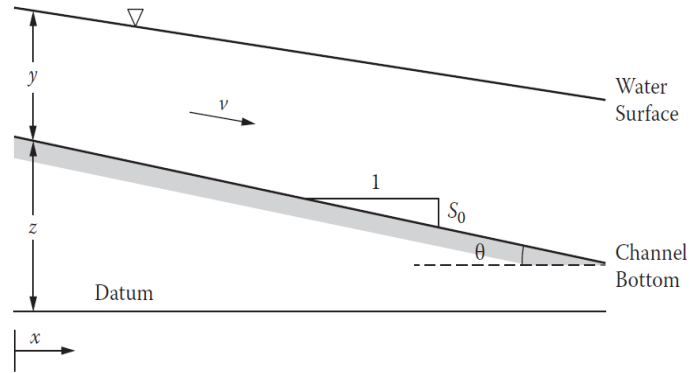


Figure 1.2. Profile view of quantities in basic open channel flow

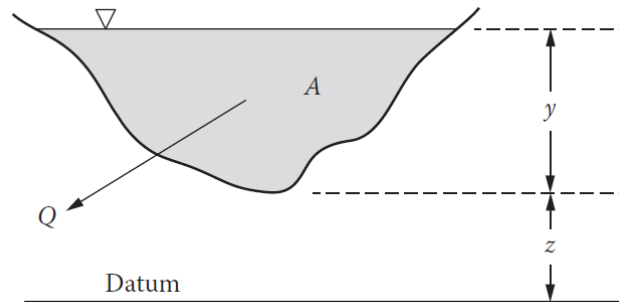


Figure 1.3. View of basic open channel flow quantities cross-section

Although it should be understood by the reader that the natural world offers an endless range of possible channel shapes, this text focuses almost exclusively on regular channel shapes: rectangular, trapezoidal, and circular. An emphasis is placed on rectangular channels, not because they are widely used, but because the mathematics of flow in a rectangular channel are as simple as possible and allow the reader to focus attention on other more pressing open channel flow phenomena. Discharge, $\dot{Q}$, is another basic open channel flow quantity. $\dot{Q}$ has a units of cubic meters per second and dimension of L^3T^{-1} . The $\dot{Q}$ also varies continuously in both the vertical and horizontal planes. Additionally, the $\dot{Q}$ is an aggregate average over the cross-sectional area of the flow. One of the fundamental properties of water is density, ρ (in ML^{-3}).

The density of water is roughly 1000 kg/m^3 [17], [18]. Closely related is the unit weight of water, γ (in FL^{-3}), which is simply the product of density and gravitational acceleration:

$$\gamma = \rho \cdot g \quad (1.1)$$

A typical value of γ is about 9810 N/m^3 in metric units or 62.4 lb/ft^3 . Density and unit weight vary slightly with temperature.

Another fundamental quality of water is its viscosity, which is reported in two different ways. The kinematic viscosity of water, ν (in L^2T^{-1}) is roughly $1.8 \times 10^{-6} \text{ m}^2/\text{s}$ at 0°C , but diminishes considerably as it warms to say, 30°C . The other way viscosity is measured and reported is as dynamic viscosity, μ (in FTL^{-2}). Dynamic and kinematic viscosity are related using equation below:

$$\mu = \rho \cdot \nu \quad (1.2)$$

The dynamic viscosity of water is approximately $1.8 \times 10^{-3} \text{ Ns/m}^2$ at 0°C , and also diminishes by a factor of two as it warms to 30°C .

In fluid hydraulics, fluids are classified as either Newtonian or non-Newtonian. A fluid with shear stress in direct proportions to its shear strain is said to have obeyed Newton's law of viscosity and is termed a Newtonian fluid. Various fluids that include air, gasses, water, kerosene, gasoline, and other oil-based liquids are Newtonian fluids. On the contrary, fluids that do not obey Newton's law of viscosity, i.e., fluids for which the shear strain and shear stress are not linearly proportional are defined as non-Newtonian fluids [19], [20], [21]. This category of fluids includes colloidal suspensions and slurries, blood, cake batter, polymer solutions, and paste. In some non-Newtonian fluids "memory" is exhibited and thus the shear stress does not only depend on the shear strain rate, but on its history also. A fluid that either partially or fully regains its shape and form after stress is applied and released is called viscoelastic [22].

Newton's 2nd law presents that constant viscosity without any elastic component and/or yield stress can be shown as:

$$\tau = \mu \gamma \quad (1.3)$$

where μ represents a constant called Newtonian viscosity, τ represents the shear stress and γ denotes the corresponding shear strain [23]. Generally, for incompressible Newtonian fluids at constant temperature, the density and shear viscosity are the determinants for stresses and velocity distributions [24].

1.1.2. Classification of open channel flows

Open channel flows are classified as follows:

Steady and Unsteady: Flow is said to be steady if the depth of flow at a particular point does not change or considered to be constant or fixed for interval of time under consideration. The flow is unsteady if depth changes with time [10], [25].

Uniform and non-uniform Flow: Space as the criterion. Open channel flow is said to be uniform if the depth and velocity of flow are the same at every section of the channel. Therefore, uniform flow can only occur in prismatic channels. Flow in channels is termed as non-uniform if the depth of flow y , changes from one section to another. A non-uniform flow is rapidly varied flow. If the depth of flow changes abruptly over a relatively short distance, e.g. Hydraulic Jump. Non-Uniform flow is gradually varied flow. If the change in depth of flow takes place gradually in a long reach of the channel [10], [24], [26].

Steady uniform flow is a flow in which the depth is constant, both with time and distance. This constitutes the fundamental type of flow in an open channel in which the gravity forces are in equilibrium with the resistance forces [27], [28].

Steady non-uniform flow is a flow in which the depth varies with distance, but not with time. The flow may be either (i) gradually varied: requires the joint application of energy and frictional resistance equations. (ii) or rapidly varied, which requires the application of energy and momentum principles [1], [10], [29].

Unsteady non uniform flow is a flow in which the depth varies with both time and distance (unsteady uniform flow is very rare). This is the most complex flow type, requiring the solution of energy, momentum, and friction equations through time. The equation below determines whether flow is said to be Laminar, turbulent or transition [1], [10], [29]. For open flow:

$$Re = \frac{\rho V D}{\mu} \quad (1.4)$$

where Re represents the dimensionless Reynolds number, V is the velocity of the fluid, μ is the dynamic viscosity of the fluid and D is the hydraulic diameter ($D = 4A/p$, where A is area of cross sectional area P is the wetted perimeter). An experimental data proved that; a flow is laminar if Re number is in the range of $500 \leq Re \leq 600$, the flow regime is transition if $500 \leq Re \leq 2000$, and the regime is turbulent is $Re > 2000$.

Subcritical, critical and supercritical flow: Gravity is a predominant force in the case of channel flow. As such depending on the relative effect of gravity the inertia forces, the channel flow may be designed as sub critical, critical, or supercritical. Determination of such flow depend on the dimensionless parameter called fraud number (Fr) which is defined as the ratio of inertia the gravity forces.

$$Fr = \frac{V}{\sqrt{gy}} \quad (1.5)$$

where V is the flow mean velocity, g is the acceleration due to gravity, y is the hydraulic depth of the channel ($y = A/T$, where T is the top width, and A is the cross sectional area). A channel flow is said to be

- (i) critical state if $Fr = 1$
- (ii) sub critical (tranquil or streaming) if $Fr < 1$, or $V\sqrt{gD}$
- (iii) supercritical or rapid flow If $Fr > 1$, or $V\sqrt{gD}$

Channels flow which occurs under free surface might either be natural such as streams and rivers, or artificial. Artificial channels consist of all man-made channels which includes irrigation, navigation canals, sewers, spillway channels, drainage ditches and culverts [30]. They are usually of regular cross-sectional shape and bed slope, which are called prismatic channels. The commonly used construction materials include steel, concrete, and earth. They are based known with well-defined surface roughness within engineering tolerances. Consequently, by applying hydraulic theories to flow in artificial channels will produce accurate results.

In contrast, natural channels are usually irregular in shape, and possess diverse materials. The surface roughness of natural channels varies with distance, time and water surface elevation. Thus, in natural channels, it is very difficult to apply hydraulic theory and obtain satisfactory results. Many applications involve man-made alterations to natural channels (e.g. flood alleviation measures and river control structures). Such applications need a proper knowledge not only of hydraulic theory, but also of the associated disciplines of hydrology, sediment transport and river morphology [30]. Different geometric properties of natural and artificial channels need to be determined for hydraulic purposes. According to Moglen [1], there are various terms used to describe channels built under different conditions such as;

Canal: a channel built on ground, i.e dug to a desired shape and slope with or without lining, commonly having a mild slope. The lining might be made of stone masonry, concrete, wood, cement, or bituminous material [10], [31].

Flume: a channel built above ground surface to convey fluid from one place to another. In the field flumes are made of wood, concrete, sheet metal or masonry. In laboratory flumes are made of metal, wood, glass or a composite of these materials [32], [33].

Chute: is a channel of steep slopes. If the change in elevation in the direction of flow occurs in a relatively short distance the channel is called a drop [34].

Culvert: is a crossing structure in running liquid which allow the flowing of liquid; culvert is relatively short and usually buried conduit commonly used for drainage purposes, as in highways and embankments. Open channel prevails whenever the culvert is flowing partially full [35].

Free-surface flow occurs over channel beds within streams, tidal estuaries, irrigation channels, rivers and hydraulic infrastructures, and when velocity of the base flow approaches critical velocity, surface waves generated by disturbances in a channel and could damages both the integrity of waterway banks as well as the hydraulic infrastructures. The free-surface flow study explored energy lost in surface waves as a significant component of the overall drag on the disturbance. Various scholarly investigation in free-surface flow are experimental studies predominantly in rectangular open channels. It is important to understand the fundamentals of the flows and it applications where there is need to protect waterway banks and hydraulic infrastructures, and reduce the wave-drag on a moving disturbance [30].

1.2. Literature Survey

In the recent years, many researchers have engaged much of their time in the study of open channel flow characteristics under different setup and conditions. It should be noted that; many studies have been reported in the literature regarding the flow behavior of different kinds of fluids in different open-channel flow setups. No available or very little information might be found in a comparative study of the influence of various obstacles on the flow characteristics of Newtonian and non-Newtonian fluids, where experimental and numerical analysis approach was used.

1.2.1. Effects of obstacles on open channel flow

An obstacle is an object in a stream that does not change the overall nature of the flow but can have finite effects on the hydraulics of the stream. The purpose here is to show that such things can and should be hydraulic structures. When water in a river or canal flows past an obstacle such as a bridge pier, piles, woody debris, or boundary roughness, there are accompanying momentum and energy losses, and the water level is different upstream and downstream of the obstacle. The problem of calculating that difference is important. There is a greater danger of flooding in the usual case of sub-critical flow when the water surface is higher upstream of an obstacle. This can be of considerable social importance. In supercritical flow, the water level is higher downstream, and the problems such as the design of stilling basins are more hydraulically than socially significant [36] [37].

Qasim et al. [38] reported that the flow in rivers, irrigation ditches, streams and channel passes obstacles such as block, dike, pier, debris material or boundary roughness can cause energy losses and as a result, the level of water around the obstacle looks different. This condition must be considered by hydraulic engineers.

Studying the use of obstacles with different shapes and roughness, Guenther et al. [39], conducted a study on the effect of continuous, staggered, and one-sided obstacle; Aal et al. [40] studied the effect of obstacle height at the edge of the spillway; Felder and Chanson [41] investigated the edge with different porosities; Al-Husseini, [42] studied on roughening of the bottom with aggregate and making an inclined edge. In general, the results of the research indicate the positive dissipation effect of obstacle on the edge.

Regarding the effect of shape and type of roughness on the bottom, systematic research was conducted by Gonzalez et al. [43]. The 3D-porous obstacle was used as an energy dissipation structure by Kordnaeij et al. [44] to control turbidity current which had a better dissipation effect than the porous plate under the same conditions due to the formation of the rotational area inside the 3D-porous obstacle.

Jasim et al. [45] reported that the impact of obstacles shape and volume in an open channel on the characteristics of a fluid, the velocity which has high impact on the characteristics of the fluid should be investigated. Stable obstacles on a surface plane provides a superior level rise than discrete obstacles when compared by Kumar et al. [46]. It was reported that the resistance force of an obstacle on a flow causes momentum loss to that flow, which takes place immediately. The momentum deficit is initially confined to the line of the obstacle, but as the turbulent flow proceeds downstream, the deficit is dispersed laterally.

Erosion in open channels is another issue that needs to be addressed and solved by using roughness elements to create more resistance in the bed. Many studies were conducted to study the hydrodynamic of open channels flow under the conditions of using roughness elements Abbas and Mohammed [47] found that the velocity magnitude was decreased with increased roughness height in the western side of Shatt al- Arab and increased in the other side.

Bora [48], studied the effect of flexible vegetation and rigidity by using PVC wires and bending it according to requirements. The study was provided different heights of wires to present like vegetation in open channels. Roughness coefficient was decreased with increased flow depth. The rigid one had more resistance than flexible one under the same flow condition. Roughness coefficient was decreased by increasing the vegetation flexibility.

Thappeta et al. [49] investigated the effect of using hemisphere boulders and cylindrical roughness elements constructed in a staggered distribution in a steep open channel. The energy loss was decreased as the density of the boulders was increased.

Mulahasan [50] who used cylindrical roughness elements with different diameters. He aimed to obtain the effect of vegetated flood-plain in open channels. For large diameter of rods, the depth of water rise quickly above the rods.

Wang [51] used five sediment roughness different in diameters and was found that the velocity distribution depends on the roughness scale and flow conditions.

The effect of obstacles, often characterized by a blockage ratio (namely DDT) or the obstacle height divided by the channel height on flame acceleration and DDT has been investigated experimentally [52], [53] and numerically [54].

Gamezo and Oran [54] noted the competing effects of high blockage ratios: larger obstacles promote non-uniform flow, which leads to fast flame acceleration, and they also weaken shocks diffracting over large obstacles. These authors also observed that the location in the channel where DDT occurs does not vary significantly for $BR = 0.31 - 0.56$ range, but it increases sharply outside of this interval for both higher and lower bars.

According to Chaulagain et al. [55] a forward-facing obstacle in a canal with no structural head drop disturbs the velocity and the pattern of flow which collides with the obstacles, resulting to estimated flow power loss. Thus, their research mainly focused on the ways of minimizing the losses due to collision and investigated the effects of insertions of obstacles along the components connecting a canal bed to a turbine platform without changing the side walls.

Ghobadian et al. [56] explained that insertion of obstacles leads to more complications in flow pattern of fluids in both open and closed channels. The authors suggested that for better understanding of effects of obstacles in fluids' flow patterns, the flow behavior around the obstacles should be studied first.

Ikani et al. [57] opined that in natural flows, bridge piers can be regarded as obstacles considering that they impede flows that are natural. According to their report, during interaction between pier and flow, the interest area are the positions immediately after the piers. this is an area where intense turbulence is experienced by the flow and exchange in momentum is experienced.

Liao and Chen [58] reported that fluid flow around an obstacle is accompanied by vortex shedding and formation, flow separation, vortex interference and other issues.

Lee et al. [59] conducted a detailed study on the DDT process in a solid obstacle-laden channel, investigated the effects of fuel type, channel diameter and obstacle blockage ratio on flame velocity, and distinguished the different flame propagation states.

Masri et al. [60] studied the interaction between the premixed flames and the obstacles of different shapes and found that the square obstacles induced the largest flame acceleration, followed by the triangular and circular obstacles.

Oh et al. [61] investigated the effect of solid obstacles with different sizes and shapes on the combustion wave acceleration and gas explosion process in a channel, and found that the obstacles could accelerate the flame propagation, but in some cases, the vortices behind the obstacles could reduce the velocity of flame propagation.

Ciccarelli et al. [62] also carried out experimental studies on flame acceleration and DDT in the obstacle-laden channels and found that the shock wave-flame interaction was a key factor affecting the flame velocity.

Han et al. [63] experimentally studied the coupling effects of obstacle position and hydrogen volume fraction on flame acceleration, and the results showed that when the hydrogen volume fraction exceeded 0.1, the maximum overpressure increased with obstacle position moving backward.

In terms of numerical simulation Gamezo et al. [64] and Oran et al. [65] investigated the influence of boundary layer and obstacles and found that both the boundary layer and obstacles could significantly affect the flame acceleration process after the reflected shock wave.

Ashley et al. [66] studied the effect of obstacle shape on the hydrogen-air DDT process and found that the interaction between the combustible gas and pressure wave could accelerate the flame.

Saeid et al. [67] numerically investigated the effect of obstacle spacing on the flame acceleration and DDT process with homogeneous and inhomogeneous hydrogen-air premix gases and found that reducing obstacle spacing would shorten the time and distance of DDT.

There are several reports on the influence of obstacles on flow of different kinds of fluids in various flow settings. Among them is Riviere et al. [68] which reported a work that studied the influence of an emerged obstacle in a supercritical open-channel flow using two work around that are of distinct form. The study focused on mass conservation and provided information on water-depth oscillation and discovered that the oscillations are caused by reversed spillage based on the associated Strouhal number.

Christodoulou [69] reported an experimental study of fluid flow in a channel with 16.5% sloping with varying kinds of artificial submerged large-scale elements of roughness. About eight obstacle's shapes which include cubes, standard baffle blocks, half-cubes, cylinders, angles, hemispheres, and vertical rods were tested in the experiment. The effective Manning's coefficient was determined based on the depth measurement, while the equivalent sand roughness was evaluated. Effects of obstacle's size, shape, and density on the flow resistance were studied.

Liao and Chen [70] used the particle image velocimetry (PIV) technique to study the flow velocity field over obstacles using aluminum oxide powder as tracers. The plots of the vertical and horizontal sections of the velocity fields as well as the streamlines were drawn and discussed. The study also investigated and compared the wake flow vortex characteristics in different flow field.

Higham and Brevis [71] undertook a study that investigated the turbulent wake changes on a confined square cylinder, induced by the introduction of 3 obstacles arrays of multiscale nature. The introduction of arrays was made around the square cylinder, downstream, and up-stream in

all cases using the same obstacles. The results showed that the shedding frequencies and energy contribution increased with the obstacles introduction through changes on the confinement.

In another report, Sharma et al. [72] studied the effects of obstacles of varying geometries with different flow rates of fluids. Obstacles are commonly used when improved intensity of turbulence in flows and better momentum exchange are needed. The work experimentally studied on flow disturbance consequences around obstacles of different shapes, sizes, and locations.

Cha et al. [73] studied the effects of obstacles of varying geometries on flow of micro fluids' inertial focusing in sinusoidal channels. The outcome showed that concave obstacles proved more effective when it comes to tuning the inertial focusing and separation of particles/ fluid than the convex obstacles. In addition, the turbulence effect induced by the trapped vehicles that serves as obstacles will also greatly improve the deflagration reaction intensity and generate more rapid flame acceleration and greater blast pressure.

Wang et al. [74] experimentally investigated the effect of obstacle shapes (rectangular, trapezoidal, and spherical) on the deflagration of premixed methane-air gas in the closed channel, and the results suggested that extremely complex changes occurred when the deflagration flames passed through the differently shaped obstacles.

Abdulmajid et al. [75] investigated the effect of separation distance of two low blockages obstacles on the acceleration of flame propagation in a vented cylindrical vessel.

Bazin et al. [76] have studied the impact of obstacles on a junction flow where two subcritical flows collide. They observed that this impact depends on the location of the obstacles and may (i) strongly modify the local velocity distribution and (ii) the extensions of a recirculation zone. Moreover, the authors observed that if an obstacle is located within a recirculation zone, the impact of the obstacle is strongly damped.

Mignot et al. [77] aimed to study the discharge distribution of different flows with obstacles located at different positions. The study showed that the discharge distribution modification relates mostly to the obstacle's location with regards to location and intersection of interface of separation and impacted strongly by the inflow Froude number, whereas the impact of water depth was very limited.

1.2.2. Effects of fluid viscosity in open channel flows (Newtonian or non-Newtonian fluids)

Fluids are classified into two namely, the Newtonian and non-Newtonian fluid. A fluid is considered Newtonian when there is a direct proportionality relation between its shear stress and the corresponding shear strain. Examples of Newtonian fluids include air, gasses, water, kerosene, gasoline and other oil-based liquids.

Newtonian fluids are governed by the Newton's law of viscosity equation that presents constant viscosity without any elastic component and/or yield stress and the governing equation

is given previously by Eq. (1.3) [23]. Generally, for the incompressible Newtonian fluids at constant temperature, the density and shear viscosity are the determinants for stresses and velocity distributions [21].

According to Aiderman, [78] fluids (whether pure or molecular solution of single-component) are considered Newtonian if they are materials of low molecular weight or when their temperature exceeds their melting point. The behavior of such fluids are characterized by a constant viscosity with respect to shear rate.

On the contrary, fluids that do not obey Newton's law of viscosity, i.e., fluids for which the shear strain and shear stress are not linearly proportional are defined as non-Newtonian fluids or nonlinear fluids [20], [21]. This category of fluids includes colloidal suspensions and slurries, blood, cake batter, polymer solutions, and paste. Other examples of non-Newtonian fluids include multi-grade oils, molten metals, printing inks, suspensions, paints, polymer solutions, molten plastics, protein solutions, blood, and ice [79]. The non-Newtonian fluids are categorized into time-dependent viscous fluids, purely viscous fluids, and viscoelastic fluids based on rheological properties analysis [80]. In some non-Newtonian fluids "memory" is exhibited and thus the shear stress does not only depend on the shear strain rate, but on its history also. A fluid that either partially or fully regains its shape and form after stress is applied and released is called viscoelastic [20]. In process equipment design such as pumps, pipelines, extruders, coaters, heat exchangers, and mixers, fluid rheology is considered important. Rheological properties can be divided into non-Newtonian and Newtonian categories. Most fluids exhibit non-Newtonian behaviors over the least coverage of a range of shear rates achievable in practice [78]. In fluid rheology, models of generalized Newtonian fluid use the assumption that non-Newtonian fluid viscosity can be elastically modeled as time-dependent are frequently utilized. Cross models and the power-law, Carreau are classical examples of such.

Poole [81] reported a study that proposed a model that predicts viscosity behavior and predicted shear and extension and also reviewed various models used in studying different fluids characteristics and suggested that a lot of the proposed models reported in the literature are not objective and highlighted the difficulties associated with their proposed model.

Among the studies reported on Newtonian and non-Newtonian fluids include the work of Maroofiazar and Farzam [82] who experimentally studied the harvesting of energy from Newtonian and non-Newtonian fluids sloshing to compare the performance of the two types of fluids in energy harvesting. The input parameters used in the study include the frequency of excitations, strength of permanent magnets, power law index of fluids, electrical resistance, and the height of non-dimensional fluid in the container.

Malekian et al., [83] reported an experimental study on the radial displacement of Newtonian-non-Newtonian liquids that are immiscible and have interfacial instability that occurs

in Hele-Shaw cells. Paraffin liquid was injected into the experiment to displace the carboxymethyl cellulose solution. A comparison between Newtonian-Newtonian and Newtonian-Non-Newtonian interfacial development indicated that Newtonian displacement showed higher finger numbers, while the fingers in the non-Newtonian system were wider and longer.

Amaratunga et al. [84] studied the vertical fluid oscillation effects on the mean velocity of settling of single spherical particles of glass released into a mixture of polymeric fluids with shear-thinning characteristics for non-zero shear rates and water. The study also discussed the shear region influence in shear-thinning non-Newtonian fluids on the rate of settling at various concentrations of oscillation of spherical particles.

Fazli et al. [85] studied the forces of interaction between fluid and particles in a particulate system of flow using the Immersed Boundary Method. The results showed that the viscosity dependence on the rate of strain in the cases of non-Newtonian fluid led to inevitable sharp viscosity change around the surface of the solid that gives rise to intermediate solution stiffness in the region of the interface.

Soligo and Rosti [86] carried out a direct simulation of non-Newtonian fluid's planar jets at low Reynold numbers in a typical Newtonian fluid's laminar conditions using a numerical approach. The fluid's characterization was performed using the relevant non-Newtonian time scale ratio of time scale defined by a parameter γ . The results showed that as the value of γ increases, a transition from a laminar flow to a turbulent flow was observed.

A novel technique for measuring viscosity was presented by Mustafa et al. [87]. A nonlinear relationship was observed between pillar depletion and fluid viscosity resulting from viscoelastic characteristics. The outcome showed that the method presented indicated a better sensitivity than using the method presented and thus, can be used as a portable viscometer for micro fluids in real-time rheological studies.

A very recent study by Hasnain et al. [88] analyzed the non-Newtonian nature of the flow of blood through the elliptical cross-sectional multi-stenosed artery. The results suggested that stenosis severity impact is reasonable on the shear stress of the wall around the stenotic region and the velocity of flow.

Romano et al. [89] simulated the fluid flow of Newtonian and non-Newtonian fluids in a vessel using CFD. The result showed that the different turbulence models used on Newtonian fluids showed a match with the velocity data determined experimentally and predicted the same mean flow.

1.2.3. Numerical studies related to open channel flow

Numerical studies on open channel flow have been reported by many different researchers studying flow characteristics under different flow conditions. Among them is the work of Jebelli

et al. [90] which simulated a canal flow using CFD code solving the Navier- Stokes equations. The effect force of outlet suction was greatly reduced on the main flow by mounting a system outlet on the water movement path. Jebelli et al. [31] simulated fluid flow in an open channel with 90° bend and studied its effective factors. The simulation investigated water flow circulation reduction after the assessment of the bend.

Keramaris et al. [91] used both the experimental and numerical approaches to study the turbulence flow of fluids in an open channel with porous and impermeable beds using the PIV and the CFD models respectively. The effects of the porous bed and the impermeable bed on the flow turbulence were also compared in the study.

Nguyen [92] developed a 3-D numerical model to simulate free flows of the surface over structures of hydraulics in rivers and natural channels. Two different methods such as front-capturing and front-tracking were used in the calculation of the profiles of the free surface.

Ben et al. [93] investigated the transportation of particulate and dissolved pollutants in channel flows that are turbulent using a numerical approach. The study presented a hydrodynamic model that is predictive in exploring the phenomena of dispersion of a pollutant pushed at an open surface over an obstacle.

Ibrahim et al. [94] performed runs using ANSYS CFX with κ - ϵ standard turbulence model and applied open channel characteristics in studying the environmental influence of hot water discharge from a thermal power plant at a cooling water source.

Natasha et al. [95] compared the characteristics of trapezoidal and rectangular open channels through the investigation of the effects of channel bottom slope, Manning coefficient, slope, and channel width using the modified Sait-Venant equation. The finite difference method with the approach of Forward Time Backward Space was employed in solving the equation numerically.

Nejad et al. [96] presented a study that investigated the counter-flow of methane-air diffusion flame into a micro/micro-open channel using time-dependent numerical analysis. The simulation was performed using the single and multi-step approach of reaction into a constant distance open channel between the fuel and the air inlet.

Mahmood et al. [97] studied the thermos-hydraulic characteristics of the fluid in a channel that's straight and isothermally heated with a flexible flow modulator under pulsating flow conditions whose bottom wall-mounted was studied using numerical analysis.

Crnković et al. [98] used a scheme algorithm and the exact conservation property in solving open channel flow equations. Several idealized tests were used in testing the behavior of the scheme and simulated a realistic flood wave propagation involving non- uniform bed slopes, friction, and strong channel width variations in Kupa River.

Mean et al. [99] reported a study on the determination of steady open channel flow profiles on water surfaces in a one-dimensional bounded domain. The study demonstrated that the viscosity solution approach indicated usefulness in describing thoroughly the properties of discontinuous and non-smooth solutions by achieving entropy conditions and momentum conservation.

Zhang et al. [100] reported on the self-similarity of population trends of span-wise rotational motion in decelerating flow in an open channel. Their study focused on the trends of the population of rotational motions span wise at different sections along the main direction of flow by using the particle imaging velocimetry measurement.

Asif et al. [101] used numerical techniques in analyzing the riverine flow behavior through the modeling of outer and central vegetation patches as rigid elements in a rectangular open channel. The influence of various configurations of vegetation patches and conditions of submergence on the flow characteristics were studied and reported using CFD simulations.

1.3. Aim of the Thesis

The aim of the study is “to investigate flow characteristics of Newtonian and non-Newtonian fluids along an open channel with defined obstacles both experimentally and numerically”.

1.4. Objectives of the Thesis

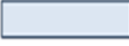
The main objectives of this thesis are:

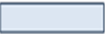
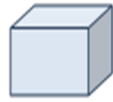
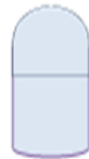
1. To find the velocity profile of both Newtonian and Non-Newtonian fluids in different positions along a channel by both experimental and numerical approaches.
2. To investigate defined obstacle shapes effects on the flow pattern.
3. To develop a methodology for CFD simulation for the flow for all cases.
4. To find out the flow depth changes near and far of the obstacles.
5. To validate the CFD outcomes by the experiment.

2. MATERIALS AND METHODS

The thesis contains both numerical analysis and experimental analysis. The open channel flow with some obstacles and slopes is analyzed for the Newtonian fluid (as water) and non-Newtonian fluid (bentonite with various mixing ratios). The experimentally and numerically analyzed cases are listed in Table 2.1.

Table 2.1. Studied cases

Cases	Method	Fluid type	Flow rate	Slope of channel	Obstacle geometry	Figure of obstacle
Case 1	Exp + Num	Water	0.0035	0	Rectangle	
Case 2	Exp + Num	Water	0.004	0	Rectangle	
Case 3	Exp + Num	Water	0.0045	0	Rectangle	
Case 4	Exp + Num	Water	0.0035	1.0156	Rectangle	
Case 5	Exp + Num	Water	0.004	1.0156	Rectangle	
Case 6	Exp + Num	Water	0.0045	1.0156	Rectangle	
Case 7	Exp + Num	Water	0.0035	1.406	Rectangle	
Case 8	Exp + Num	Water	0.004	1.406	Rectangle	
Case 9	Exp + Num	Water	0.0045	1.406	Rectangle	
Case 10	Exp + Num	Water	0.0035	0	Rectangular arc	
Case 11	Exp + Num	Water	0.004	0	Rectangular arc	
Case 12	Exp + Num	Water	0.0045	0	Rectangular arc	
Case 13	Exp + Num	Water	0.0035	1.0156	Rectangular arc	
Case 14	Exp + Num	Water	0.004	1.0156	Rectangular arc	
Case 15	Exp + Num	Water	0.0045	1.0156	Rectangular arc	
Case 16	Exp + Num	Water	0.0035	1.406	Rectangular arc	
Case 17	Exp + Num	Water	0.004	1.406	Rectangular arc	
Case 18	Exp + Num	Water	0.0045	1.406	Rectangular arc	
Case 19	Exp + Num	Water	0.0035	0	Cube	
Case 20	Exp + Num	Water	0.004	0	Cube	
Case 21	Exp + Num	Water	0.0045	0	Cube	
Case 22	Exp + Num	Water	0.0035	1.0156	Cube	
Case 23	Exp + Num	Water	0.004	1.0156	Cube	
Case 24	Exp + Num	Water	0.0045	1.0156	Cube	
Case 25	Exp + Num	Water	0.0035	1.406	Cube	
Case 26	Exp + Num	Water	0.004	1.406	Cube	
Case 27	Exp + Num	Water	0.0045	1.406	Cube	
Case 28	Exp. only	Water	0.0035	0	Cylindrical	
Case 29	Exp. only	Water	0.004	0	Cylindrical	
Case 30	Exp. only	Water	0.0045	0	Cylindrical	
Case 31	Exp. only	Water	0.0035	1.0156	Cylindrical	
Case 32	Exp. only	Water	0.004	1.0156	Cylindrical	
Case 33	Exp. only	Water	0.0045	1.0156	Cylindrical	
Case 34	Exp. only	Water	0.0035	1.406	Cylindrical	
Case 35	Exp. only	Water	0.004	1.406	Cylindrical	
Case 36	Exp. only	Water	0.0045	1.406	Cylindrical	

Case 37	Exp. only	Water	0.0035	0	Elliptical	
Case 38	Exp. only	Water	0.004	0	Elliptical	
Case 39	Exp. only	Water	0.0045	0	Elliptical	
Case 40	Exp. only	Water	0.0035	1.0156	Elliptical	
Case 41	Exp. only	Water	0.004	1.0156	Elliptical	
Case 42	Exp. only	Water	0.0045	1.0156	Elliptical	
Case 43	Exp. only	Water	0.0035	1.406	Elliptical	
Case 44	Exp. only	Water	0.004	1.406	Elliptical	
Case 45	Exp. only	Water	0.0045	1.406	Elliptical	
Case 46	Exp. + Num.	Bentonite 70%	0.0035	0	Rectangle	
Case 47	Exp. + Num.	Bentonite 70%	0.004	0	Rectangle	
Case 48	Exp. + Num.	Bentonite 70%	0.0045	0	Rectangle	
Case 49	Exp. + Num.	Bentonite 70%	0.0035	1.0156	Rectangle	
Case 50	Exp. + Num.	Bentonite 70%	0.004	1.0156	Rectangle	
Case 51	Exp. + Num.	Bentonite 70%	0.0045	1.0156	Rectangle	
Case 52	Exp. + Num.	Bentonite 70%	0.0035	1.406	Rectangle	
Case 53	Exp. + Num.	Bentonite 70%	0.004	1.406	Rectangle	
Case 54	Exp. + Num.	Bentonite 70%	0.0045	1.406	Rectangle	
Case 55	Exp. + Num.	Bentonite 70%	0.0035	0	Rectangular arc	
Case 56	Exp. + Num.	Bentonite 70%	0.004	0	Rectangular arc	
Case 57	Exp. + Num.	Bentonite 70%	0.0045	0	Rectangular arc	
Case 58	Exp. + Num.	Bentonite 70%	0.0035	1.0156	Rectangular arc	
Case 59	Exp. + Num.	Bentonite 70%	0.004	1.0156	Rectangular arc	
Case 60	Exp. + Num.	Bentonite 70%	0.0045	1.0156	Rectangular arc	
Case 61	Exp. + Num.	Bentonite 70%	0.0035	1.406	Rectangular arc	
Case 62	Exp. + Num.	Bentonite 70%	0.004	1.406	Rectangular arc	
Case 63	Exp. + Num.	Bentonite 70%	0.0045	1.406	Rectangular arc	
Case 64	Exp. + Num.	Bentonite 70%	0.0035	0	Cube	
Case 65	Exp. + Num.	Bentonite 70%	0.004	0	Cube	
Case 66	Exp. + Num.	Bentonite 70%	0.0045	0	Cube	
Case 67	Exp. + Num.	Bentonite 70%	0.0035	1.0156	Cube	
Case 68	Exp. + Num.	Bentonite 70%	0.004	1.0156	Cube	
Case 69	Exp. + Num.	Bentonite 70%	0.0045	1.0156	Cube	
Case 70	Exp. + Num.	Bentonite 70%	0.0035	1.406	Cube	
Case 71	Exp. + Num.	Bentonite 70%	0.004	1.406	Cube	
Case 72	Exp. + Num.	Bentonite 70%	0.0045	1.406	Cube	
Case 73	Exp. only	Bentonite 70%	0.0035	0	Cylindrical	
Case 74	Exp. only	Bentonite 70%	0.004	0	Cylindrical	
Case 75	Exp. only	Bentonite 70%	0.0045	0	Cylindrical	
Case 76	Exp. only	Bentonite 70%	0.0035	1.0156	Cylindrical	
Case 77	Exp. only	Bentonite 70%	0.004	1.0156	Cylindrical	
Case 78	Exp. only	Bentonite 70%	0.0045	1.0156	Cylindrical	
Case 79	Exp. only	Bentonite 70%	0.0035	1.406	Cylindrical	
Case 80	Exp. only	Bentonite 70%	0.004	1.406	Cylindrical	
Case 81	Exp. only	Bentonite 70%	0.0045	1.406	Cylindrical	
Case 82	Exp. only	Bentonite 70%	0.0035	0	Elliptical	
Case 83	Exp. only	Bentonite 70%	0.004	0	Elliptical	
Case 84	Exp. only	Bentonite 70%	0.0045	0	Elliptical	
Case 85	Exp. only	Bentonite 70%	0.0035	1.0156	Elliptical	
Case 86	Exp. only	Bentonite 70%	0.004	1.0156	Elliptical	
Case 87	Exp. only	Bentonite 70%	0.0045	1.0156	Elliptical	
Case 88	Exp. only	Bentonite 70%	0.0035	1.406	Elliptical	
Case 89	Exp. only	Bentonite 70%	0.004	1.406	Elliptical	
Case 90	Exp. only	Bentonite 70%	0.0045	1.406	Elliptical	

Case 91	Exp. + Num.	Bentonite 26%	0.0035	0	Rectangular arc	
Case 92	Exp. + Num.	Bentonite 26%	0.004	0	Rectangular arc	
Case 93	Exp. + Num.	Bentonite 26%	0.0045	0	Rectangular arc	
Case 94	Exp. + Num.	Bentonite 26%	0.0035	1.0156	Rectangular arc	
Case 95	Exp. + Num.	Bentonite 26%	0.004	1.0156	Rectangular arc	
Case 96	Exp. + Num.	Bentonite 26%	0.0045	1.0156	Rectangular arc	
Case 97	Exp. + Num.	Bentonite 26%	0.0035	1.406	Rectangular arc	
Case 98	Exp. + Num.	Bentonite 26%	0.004	1.406	Rectangular arc	
Case 99	Exp. + Num.	Bentonite 26%	0.0035	0	Rectangular arc	
Case 100	Exp. + Num.	Bentonite 11%	0.0035	0	Rectangular arc	
Case 101	Exp. + Num.	Bentonite 11%	0.004	0	Rectangular arc	
Case 102	Exp. + Num.	Bentonite 11%	0.0045	0	Rectangular arc	
Case 103	Exp. + Num.	Bentonite 11%	0.0035	1.0156	Rectangular arc	
Case 104	Exp. + Num.	Bentonite 11%	0.004	1.0156	Rectangular arc	
Case 105	Exp. + Num.	Bentonite 11%	0.0045	1.0156	Rectangular arc	
Case 106	Exp. + Num.	Bentonite 11%	0.0035	1.406	Rectangular arc	
Case 107	Exp. + Num.	Bentonite 11%	0.004	1.406	Rectangular arc	
Case 108	Exp. + Num.	Bentonite 11%	0.0035	0	Rectangular arc	

2.1. Experimental Analysis

The experiments were conducted within a controlled environment designed specifically for fluid dynamics, specifically in an open channel containing obstacles of different sizes and shapes. This section provides an in-depth account of the materials used and the experimental setup, including a comprehensive description of the laboratory model, and an illustration of the channel design along with obstacle configurations. Furthermore, it delves into the complexities of data collection, outlining the methodology for measuring velocity and depth across multiple channel segments, as well as illustrating variations in discharge rates and channel inclination. In addition, the section outlines the data analysis process and provides insight into the general approach to data processing techniques, with an emphasis on the use of Excel for comprehensive analysis and visualization purposes.

2.1.1. Setup

To comprehensively evaluate the effect of regular obstructions on the dynamics of water flow within an open channel, a specialized infrastructure has been carefully designed. The experimental setup is viewed with a real photo in Fig. 2.1 and with a schematic picture created by Solid works in Fig. 2.2. The channel itself, which is 400 cm long, 30 cm wide, and 40 cm deep, has been meticulously crafted using a sturdy iron frame. To ensure optimal visibility and accuracy in measurements, the sides and bottom of the channel are meticulously crafted from strong glass sheets, ensuring transparency throughout the experimental process.

To facilitate control of water flow, a powerful 9 kW pump, coupled with 3 inch pipes, is intricately integrated into the system. This state-of-the-art setup facilitated the seamless transfer of water from a dedicated 1 m³ supply tank to a small tank strategically placed at the starting

point of the channel. The precise arrangement of components is precisely designed to minimize turbulence and enhance laminar flow, thus enhancing the accuracy of experimental results.

Furthermore, the channel was designed with adaptability in mind, providing the flexibility to adjust the longitudinal slope as dictated by experimental requirements. To enable accurate measurement of different discharge rates, the channel is equipped with a regulator and a magnetic flow meter. These instruments play a pivotal role in accurately controlling and measuring flow rates during an experiment, ensuring data integrity and reliability.



Figure 2.1. Experimental setup of the open channel in the laboratory



Figure 2.2. Schematic view of the open channel system

In terms of obstacle placement, each type of systemic obstacle was distributed regularly and in a uniform manner within the central third of the channel, as shown in Fig. 2.1. This planned arrangement ensures consistency in the distribution of obstacles, both in shape and spacing, thus facilitating uniform experimental conditions.

The experimental protocol involved making the depth and velocity measurements at 3 specific points per cross-section along a total of 15 sections distributed throughout the channel. These measurements were carried out precisely, taking into account the density of obstacles within each specific area.

Focusing specifically on the effects of obstruction, measurements were taken systematically at intervals along three distinct paths within each designated section. This careful approach ensures comprehensive data collection across various flow conditions and obstacle configurations, facilitating robust analysis of experimental results.

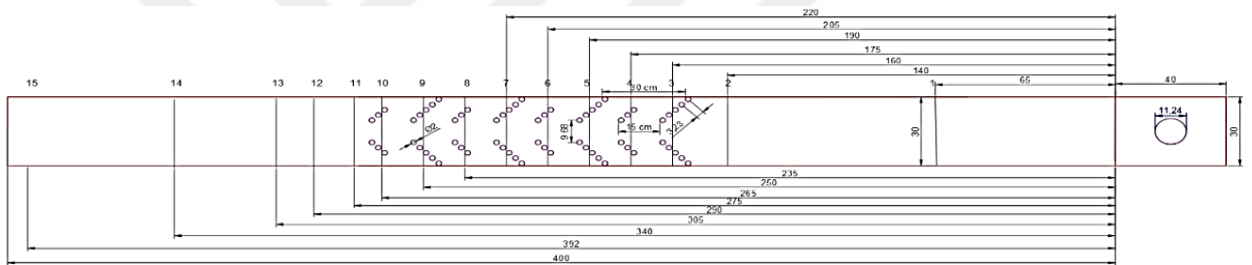
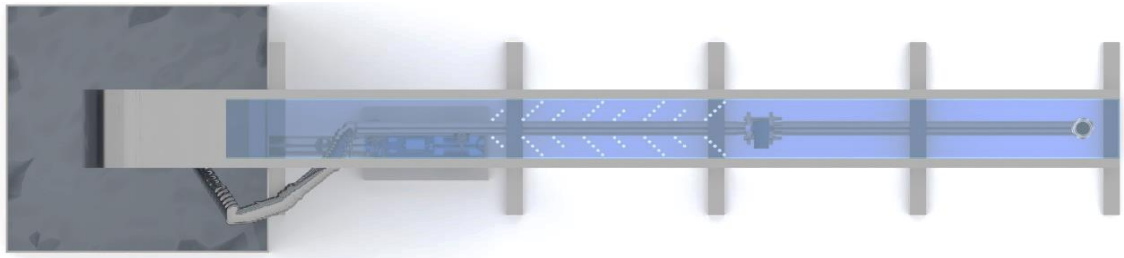
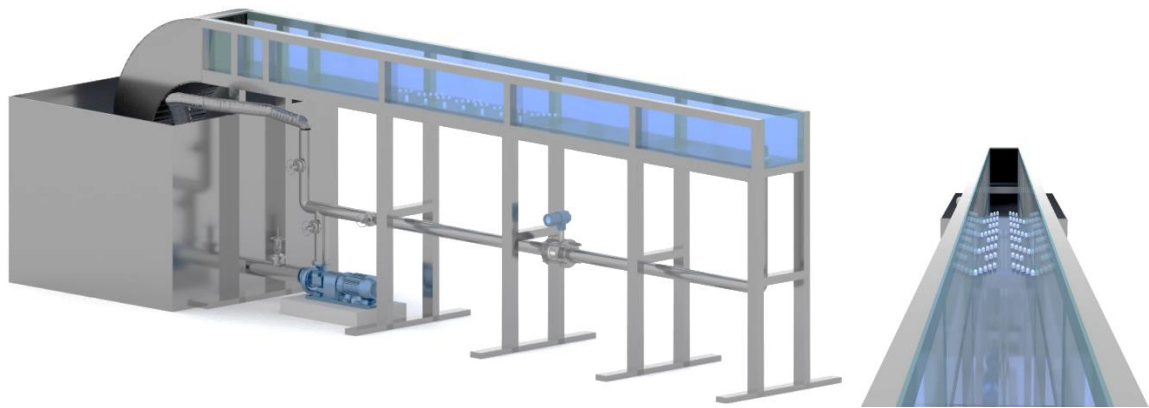
2.1.2. Obstacles and their arrangements

The experimental setup focuses on precisely positioning obstacles along the channel to significantly demonstrate their influence on fluid flow. To cover a wide range of situations, the selection of these obstacle shapes is based on requirements to include the most relevant common shapes. Five different shapes were used: rectangle, cube, cylindrical, arc, and curved rectangle. These shapes are precisely designed to ensure consistent obstruction of fluid flow within the canal.

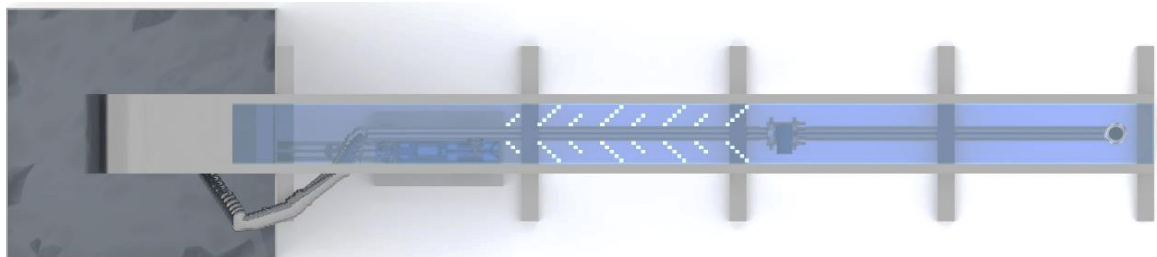
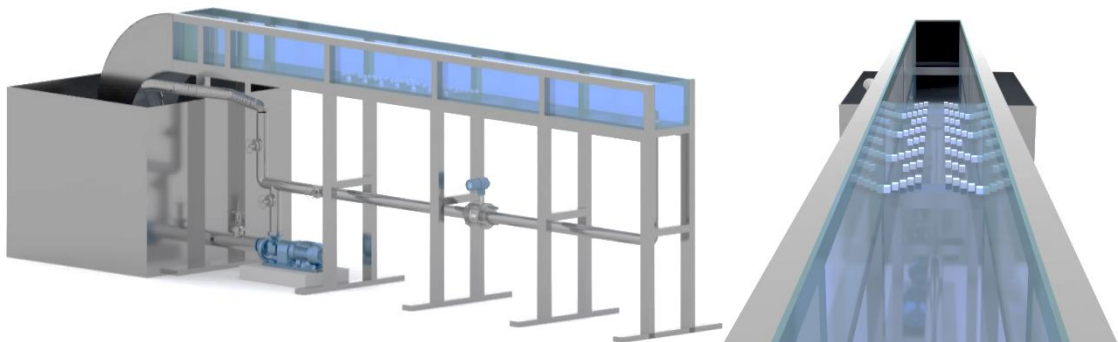
Rectangular and cubic baffles have well-defined edges, which impart unique effects on fluid flow compared to more defined fluid shapes such as cylindrical, crescent-shaped, and curved rectangular baffles. This diversity in shape allows exploration of how different geometries affect fluid dynamics.

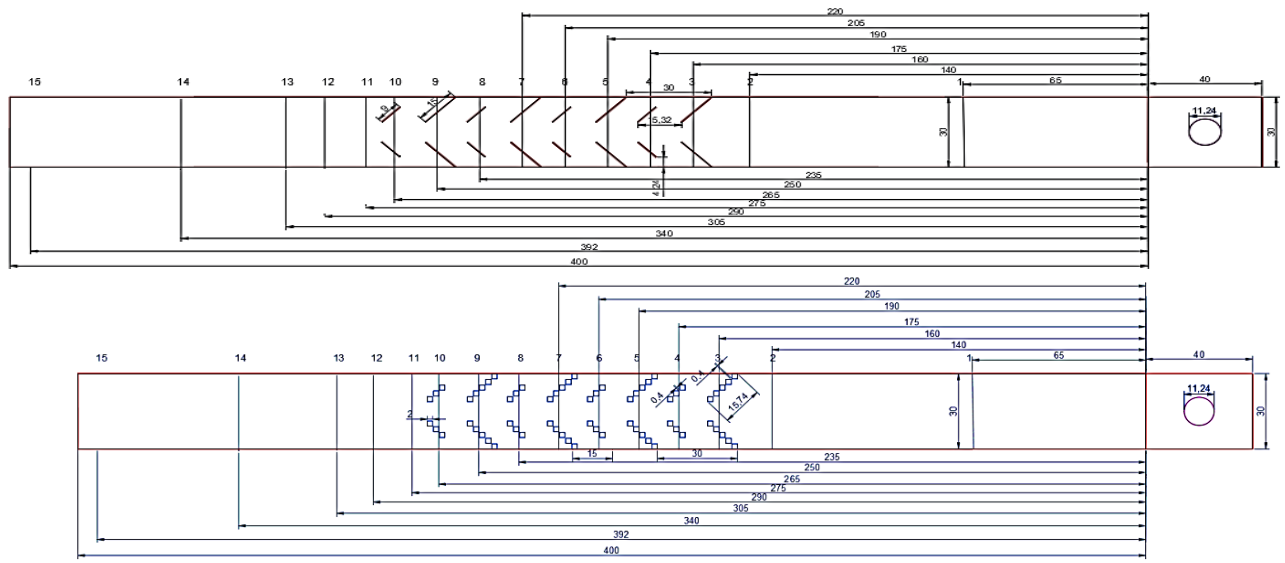
The placement of these obstructions is strategic, with varying durations along the channel path, thus presenting manageable challenges for flowing fluids. It is worth noting that a set of obstacles 2 m long was installed in the center of the channel to ensure that the fluid flow is completely stable before it is encountered.

The arrangement and orientation of these obstacles are visually depicted in Fig. 2.3 (a-d) which illustrate the alignment and directions of obstacles in the experimental setup. This comprehensive approach ensures a comprehensive investigation, with each set of obstacles subject to three flow rates for each specific slope, providing comprehensive data for analysis and interpretation. Furthermore Fig. 2.4 is plotted so show the arrangement procedure of the obstacles. The measurement points are also shown in Fig. 2.4 with yellow dots. The general view of the obstacles in the channel is shown in Fig. 2.4, one shows the obstacles closer, one shows from the top view.

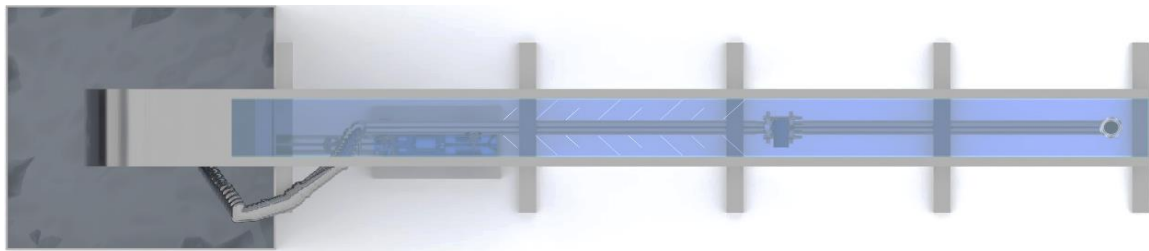
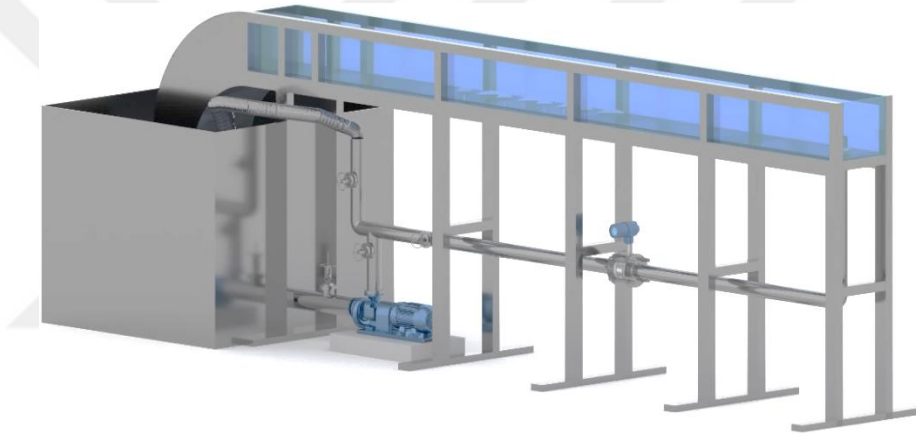


a)

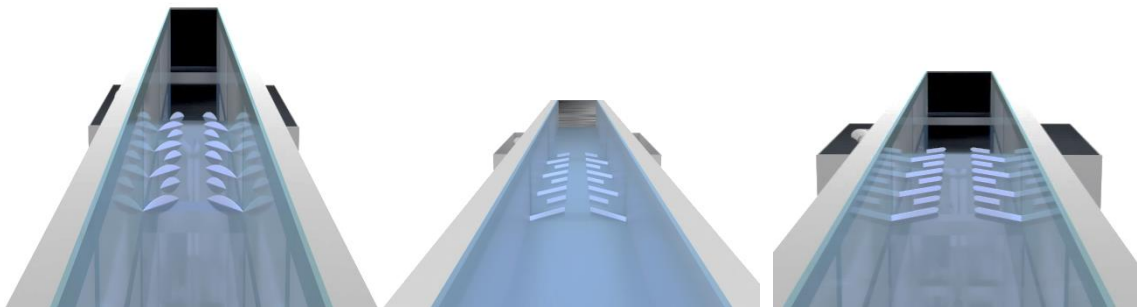




b)



c)



d)

Figure 2.3. Layout of obstacles; **a)** cube, **b)** short cylindrical, **c)** arc plate, **d)** rectangular plate,

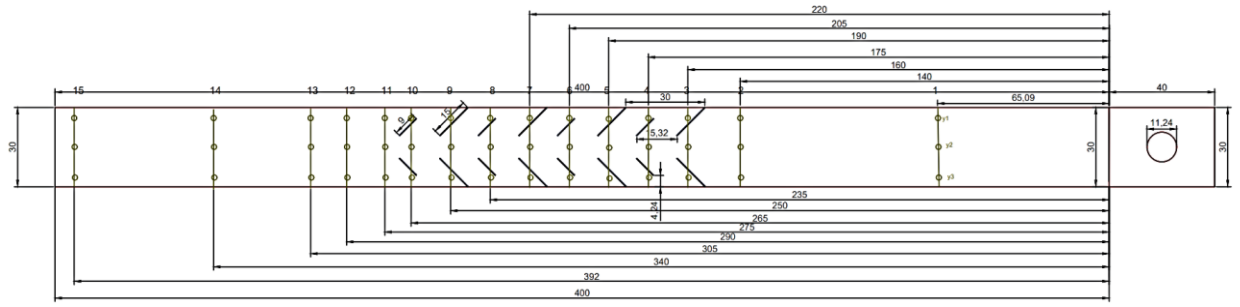


Figure 2.4. Arrangement of measurements area

2.1.3. Fluids selection and preparations

The experiment was conducted using two distinct types of fluid: a Newtonian fluid, which is water, and a non-Newtonian fluid, a carefully prepared mixture of water and bentonite. Precise preparation of a non-Newtonian fluid involves mixing water and bentonite to create a mixture that closely mimics real-world scenarios where non-Newtonian behavior is observed in fluids. This deliberate choice was made to facilitate a comprehensive examination of how variations in viscosity, influenced by shear rate, affect fluid flow within a channel obstructed by obstructions. Many experiments were conducted to reach the appropriate mixture with the most common density. The experiments started with the mixing ratios as shown in Table 2.2.

Table 2.2. Bentonite concentration ratios and properties

Fluid	Density ρ (kg/m ³)	Consistency index k (kg·s ⁿ⁻²)	Power law index n	Yield stress threshold τ_y (Pa)	Critical shear rate SR (1/s)
Bentonite suspension 3%	1014	0.003	1	1	0.1
Bentonite suspension 4%	1025	0.006	1	4.4	0.1
Bentonite suspension 6%	1033	0.006	1	12.7	0.1

Furthermore, the viscosity of the non-Newtonian fluid was determined experimentally by measuring its viscosity for various revolutions per minute (RPM), which reflects the variation in velocity during flow in the open channel. This approach ensures that the rheological properties of the non-Newtonian fluid accurately represent its behavior under different flow conditions, thus enhancing the accuracy of the experiment results. The selection of these specific fluids is crucial because it allows a deeper understanding of the complex interactions between fluid properties and barrier configurations. By incorporating both Newtonian and non-Newtonian fluids, the experiment aims to shed light on how different rheological behaviors affect flow dynamics in obstacle-laden channels. This comprehensive approach enhances the ability of the experiment to

simulate real-world conditions and provides valuable insights into the behavior of fluid systems in practical applications. Fig. 2.5 and Fig. 2.6 depict the viscometer used to determine the viscosity in the experiment. This essential device plays an essential role in accurately measuring the viscosity of fluids under various conditions. Through the use of this specialized equipment, researchers can obtain accurate data on the viscosity of both Newtonian and non-Newtonian fluids, thus enabling a comprehensive understanding of their rheological behavior. Viscometer serves as a valuable tool in ensuring the reliability and accuracy of experimental results, ultimately contributing to the advancement of knowledge in fluid dynamics and related fields.



Figure 2.5. Viscometer used for viscosity determination.



a)

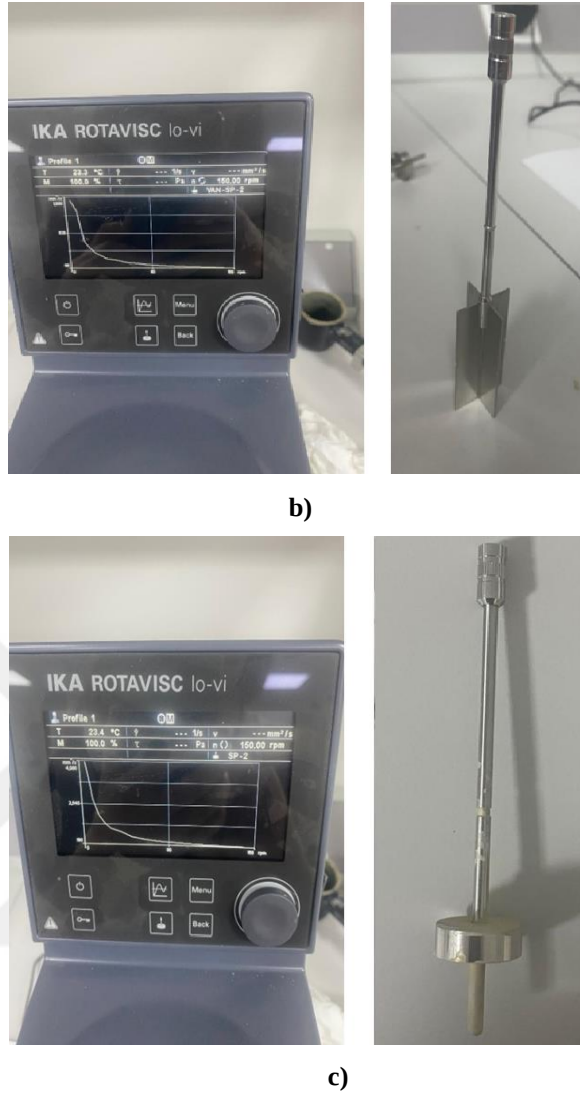


Figure 2.6. Viscometer with **a)** long cylindrical spindle, **b)** spindle with four perpendicular plates, **c)** very short cylindrical spindle

Preparation of non-Newtonian bentonite mixtures involves careful mixing of bentonite clay with water to achieve different concentration ratios, specifically 14 g bentonite in 1000 g water, 26 g bentonite in 1000 g water, and 70 g bentonite in 1000 g water. These mixtures are used to study flow dynamics in open channels containing obstacles, providing valuable insights into the behavior of fluids in complex environments. Table 2.3 presents the mixing ratios.

Table 2.3. Mixing ratios and densities of considered mixtures

Mixture	Mixing Ratio	Density g/L	Density PPG
A	11.8 g/L	1007.38	8.41
B	26 g/L	1013.53	8.4123
C	70 g/L	1039.19	8.625

Bentonite clay: A soft, absorbent clay composed mostly of montmorillonite.

Water: Normal pure water is used in this process. It is preferable to use deionized water to avoid any impurities that may affect the properties of the mixture.

Mixing Bowls: Graduated bowls for accurate measurement of water and bentonite.

Stirring devices: Mechanical stirrers or magnetic stirrers to ensure precise mixing.

Measuring Ingredients: Using an accurate scale, weigh the bentonite and water to each desired concentration.

For the three specified ratios:

- ✓ 11g bentonite in 1000g water: Light concentration, resulting in a more flexible mixture.
- ✓ 26 grams of bentonite in 1000 grams of water: medium concentration to form a fairly thick mixture.
- ✓ 70 g of bentonite in 1000 g of water: high concentration, resulting in a noticeably thick and sticky mixture.

Initial mixing: Gradually add the bentonite to the water, stirring constantly to prevent clumping. It is necessary to add the bentonite slowly to allow it to distribute evenly in the water.

Humidification period: Allow the mixture to moisturize for a specified period, usually several hours. This step ensures that the bentonite particles are fully swollen and uniformly distributed.

Final mixing: After the humidification period, perform a thorough final mixing to obtain a homogeneous non-Newtonian fluid. Mechanical stirrers are particularly effective at this stage to ensure consistency.

2.1.4. Density measurements

The density of prepared mixtures is critical for characterizing fluid properties and ensuring consistency across experiments. A laboratory device shown in Fig. 2.7, usually a pycnometer or digital densitometer, is used to accurately measure the density of each mixture. This measurement is vital because it affects flow behavior and interaction with obstacles in open channel studies. Specific gravity of bentonite mixtures are plotted in Fig. 2.8. additionally the cup for the density measurements are given in Fig. 2.9.

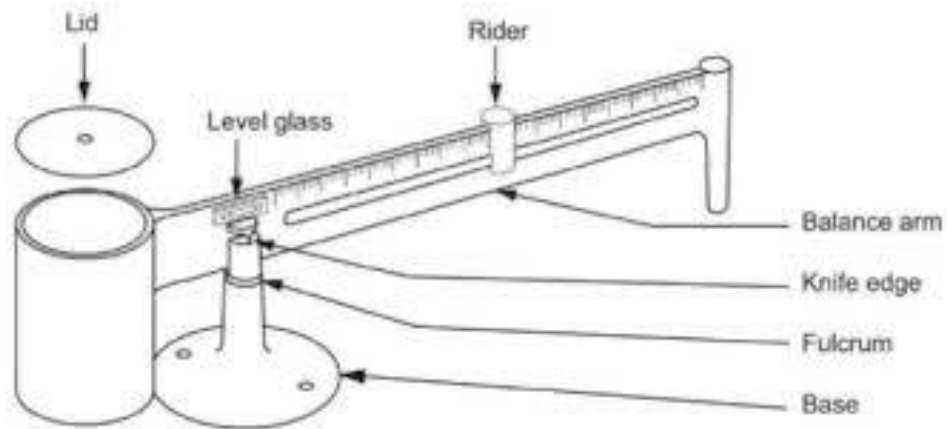


Figure 2.7. Density determination apparatus

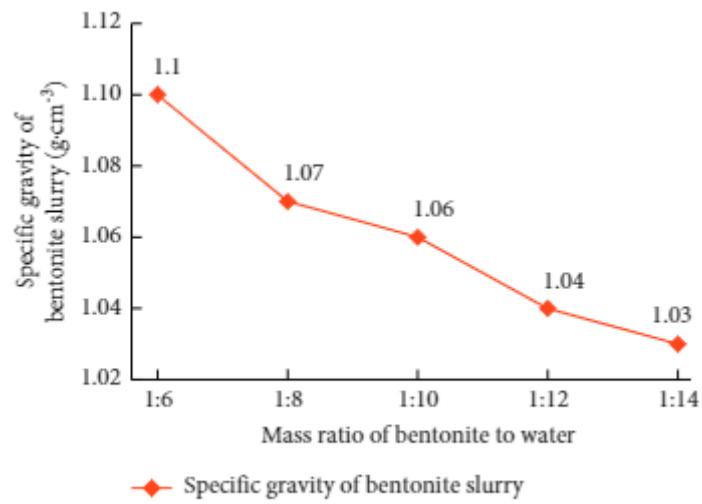


Figure 2.8. Specific gravity of bentonite mixtures [20]



Figure 2.9. The density measurement cup

2.1.5. Viscosity measurements

The viscosity of bentonite mixtures, a fundamental property that determines their non-Newtonian behavior, is measured using a viscometer. Different manifolds are used to capture viscosities at different shear rates, providing a comprehensive overview of the rheological properties of the fluid.

Long cylindrical spindle:

The viscosity (V) values at varying revolutions (rpm) show shear thinning behavior, where the viscosity decreases with increasing shear rate.

Ultra short cylindrical spindle:

This mandrel provides viscosity readings for lower shear rates, highlighting the fluid's resistance to flow under minimal agitation.

Spindle with four perpendicular plates:

This mandrel is particularly useful for capturing viscosities at higher shear rates, demonstrating the non-Newtonian properties of the bentonite mixture.

Together, these measurements provide a detailed understanding of how bentonite concentrations affect viscosity and thus flow dynamics in practical applications involving open channels with obstructions. The systematic decrease in viscosity with increasing rpm across all spindle types confirms the non-Newtonian and shear-thinning nature of bentonite-water mixtures, which is critical for modeling and predicting their behavior in various engineering and environmental contexts. Fig. 2.10 shows the variation of kinematic viscosity for different viscometer spindles. Table 2.4 shows kinematic viscosities.

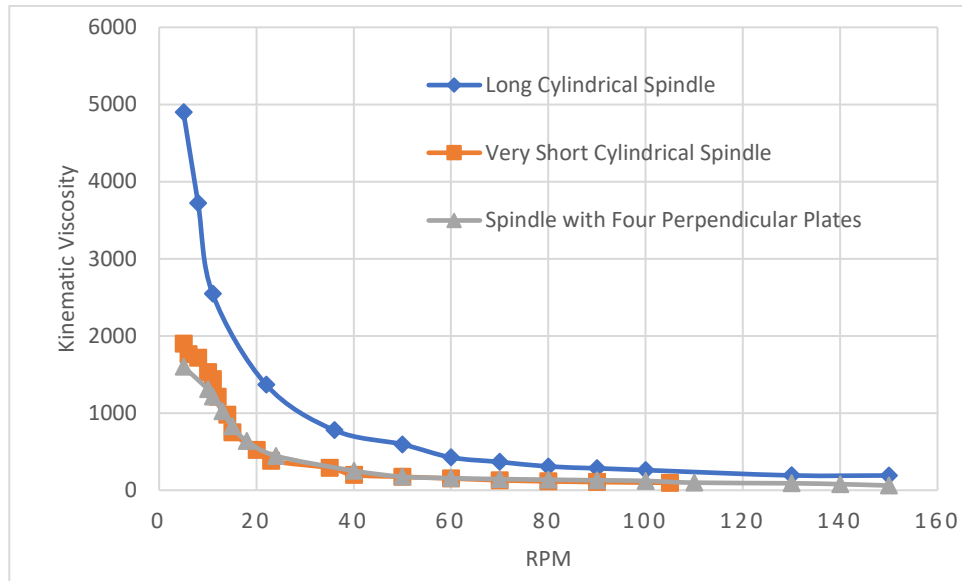


Figure 2.10. Variation of kinematic viscosity for different viscometer spindles

Table 2.4. Kinematic viscosity for different viscometer spindles

Very Short Cylindrical Spindle		Spindle with Four Perpendicular Plates		Long Cylindrical Spindle	
RPM	ν (mm ² /sec)	RPM	ν (mm ² /sec)	RPM	ν (mm ² /sec)
5	1900	5	1600	5	4900
6	1762	10	1311.25	8	3722.5
8	1716	11	1215	11	2545
10	1532	13	1022.5	22	1367.5
11	1440	15	830	36	778.75
12	1210	18	637.5	50	594.375
14	980	24	445	60	425.5
15	750	40	252.5	70	366.625
20	520	50	175.5	80	307.75
23	382	60	156.25	90	284.2
35	290	70	144.7	100	260.65
40	198	80	137	130	192
50	175	90	129	150	191
60	152	100	121		
70	129	110	98		
80	115	130	90		
90	106	140	79		
105	96	150	61		
130	87				
145	78				
150	62				
150	62				

2.1.6. Procedure and data reduction

The experiment aims to comprehensively analyze the flow characteristics of both a Newtonian fluid (water) and a non-Newtonian fluid (a mixture of water and bentonite). This investigation was performed in a laboratory setting using an open channel system. The system includes pumping rates set at 0.0035, 0.0040, and 0.0045, corresponding to varying flow intensities. In addition, each flow rate was tested under inclined slope angles of 0° , 1.0156° , and 1.406° allowing flow behavior under different hydraulic gradients to be examined.

The experimental procedure began with the lowest flow rates and lowest channel slope, with the slope gradually increasing for each subsequent flow rate. This methodological approach ensures a comprehensive exploration of flow dynamics across a range of conditions. Throughout the experiment, strict measures were taken to maintain the temperature within normal ambient conditions, thus minimizing potential confounding factors.

In the measurement process, depth and velocity were accurately evaluated at three points across fifteen sections along the open channel. It is worth noting that denser measurements were performed within the area extending from 65 cm to 250 cm, where all the obstacles studied were strategically arranged. This detailed measurement strategy aims to capture subtle variations in flow characteristics, especially in areas directly affected by obstacle placement. Flume tests were conducted within 4 m rectangular culverts, specially designed and constructed at Tishk International University in Erbil, Iraq. This flume feature can tilt from the horizontal position, providing an adaptable range of channel slope configurations to effectively accommodate the required experimental parameters. The measurement devices are shown in Fig. 2.11 and Fig. 2.12.



Figure 2.11. Digital current meter

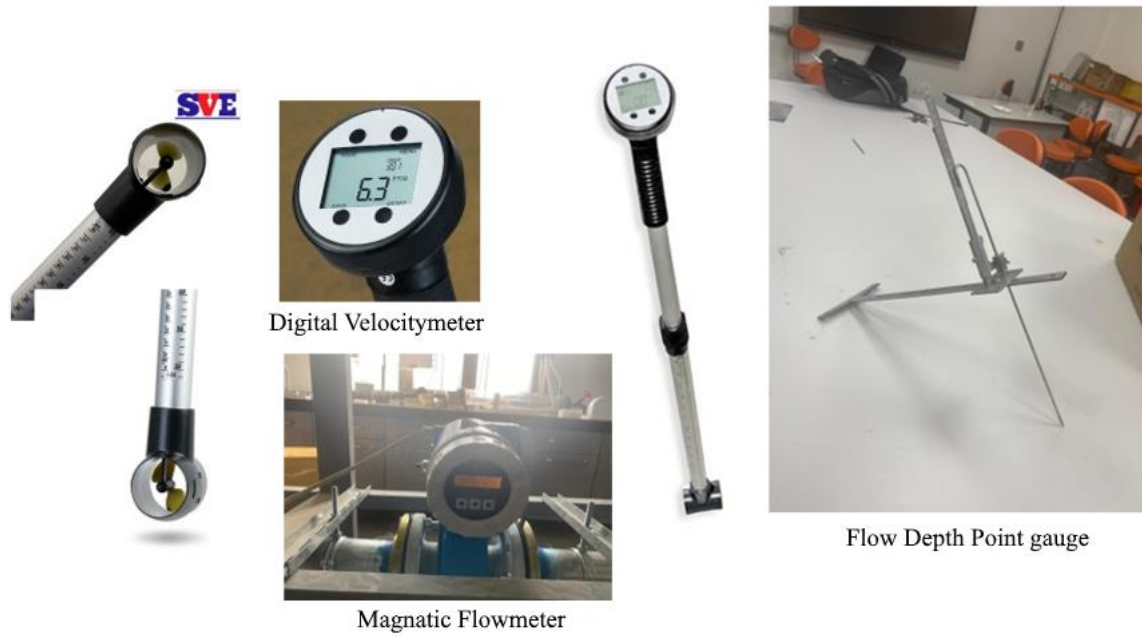


Figure 2.12. Digital velocity meter, flow depth point gauge, and magnetic flowmeter for water measurements.

Flow within the flume is facilitated by two primary pumps: a 100 mm progressive bore positive displacement pump and a 4 x 3 centrifugal slurry pump. These pumps, equipped with variable velocity drives, allow precise control of flow rates. Flow rates were monitored using an electromagnetic flow meter, ensuring accurate measurement and control throughout the experiment.

It is worth noting that the maximum discharge capacity of the flume was set at 0.0045 cubic meters per second, which provides sufficient capacity to accommodate the experimental conditions and ensure reliable data.

When conducting measurements and documentation for the canal study, a carefully structured and planned approach was implemented to ensure accuracy and comprehensiveness. The channel, which is 4 meters long, is precisely divided into fifteen measuring stations. These stations were strategically distributed at unequal intervals to appropriately capture the nuances of flow dynamics found along the channel. Furthermore, the obstructions were strategically placed across eight distinct sections, extending from 140 meters to 285 meters from the canal entrance. These sections were carefully selected to represent the different hydraulic challenges, with equal spacing between each obstacle.

The primary focus of the measurements was to capture depth and velocity at three distinct points across each of the 15 sections along the channel. This systematic approach has allowed a precise understanding of the channel flow characteristics and their changes along its entire length.

All collected data were carefully recorded and indexed in spreadsheets using Excel. This choice facilitated the smooth organization and analysis of the data set. Each measuring station and obstacle section is meticulously tabulated in an Excel file, ensuring a coherent, easy-to-navigate data set. Furthermore, to facilitate seamless comparison with results derived from digital models, the collected data has been carefully formatted to align with the model outputs. This precise alignment allowed for easy integration and validation of the digital model against real-world observations. The structured layout and presentation of data within the Excel file, represented by Fig. 2.13 in the documentation, was a clear and comprehensive illustration of how data was systematically organized and accurately recorded. This comprehensive approach to measurement and documentation ensures the reliability and validity of the study results, thus enabling informed decision-making and facilitating further improvements to the numerical model.

In addition to the comprehensive depth and velocity measurements made across the 15 sections of the channel, it is important to highlight that the study also took into account three distinct flow rates: $\dot{Q} = 0.0035 \text{ m}^3/\text{s}$, $0.004 \text{ m}^3/\text{s}$, and $0.0045 \text{ m}^3/\text{s}$. Each flow rate was carefully examined across three different longitudinal slopes to capture diverse hydraulic conditions. This comprehensive approach facilitated a comprehensive analysis of the effects of varying flow rates and slopes on channel flow dynamics and behavior. By performing measurements under specific flow rate conditions, the study ensured representative sampling of different flow regimes, thus enhancing the robustness of the results and enabling more accurate predictions of flow patterns and sediment transport within the channel.

To comprehensively investigate fluid flow dynamics under diverse conditions, especially around obstacles, a careful approach involves coupling video recordings with precise measurements. This integrated methodology allows a detailed examination of phenomena such as fluctuation of streamlines and effects caused by obstructive elements within the flow path. Notably, observations reveal distinct occurrences such as flow separation and vortex formation, which appear prominently in scenarios involving water flow and to a lesser extent in the context of non-Newtonian fluids. To expand on this, the use of video recordings serves as a valuable tool in capturing the complex behavior of fluid flow. By combining visual data with precise measurements, researchers can gain insight into the precise interactions occurring within a flow system. This approach not only facilitates the identification of phenomena such as streamlining fluctuations but also allows the observation of complex flow patterns arising from the presence of obstacles.

ensures that every piece of data is accurately captured and organized, facilitating comprehensive analysis and interpretation. Furthermore, to complement our experimental observations, numerical modeling using sophisticated software such as CFX has been used. This numerical modeling allows us to simulate the flow dynamics within the channel by incorporating boundary conditions. These simulations generate numerical results that provide valuable insights into the behavior of the fluid under different conditions. It is important to note that these numerical results are not independent; It is an integral part of our research methodology. To ensure the accuracy and reliability of our numerical simulations, they are systematically compared with our experimental results. This comparative analysis is a crucial step in validating the numerical model, ensuring that it accurately represents real-world flow behavior. By comparing the numerical results with our experimental data, It is important that any inconsistencies or; areas where this model might need improvement, could be identified. This iterative process of comparison and validation enhances the credibility of our findings and strengthens the overall integrity of our study.

However, the combination of careful documentation of experimental measurements in Excel files and the use of numerical modeling through software such as CFX forms a powerful framework for our research. This integrated approach not only allows us to comprehensively analyze the experimental data but also allows us to validate our numerical simulations, ensuring the accuracy and reliability of our findings.

2.2. Numerical Analysis

2.2.1. Physical model

Computational Fluid Dynamics (CFD) has emerged as a powerful tool in fluid dynamics analysis and simulation, providing deep insights into complex flow phenomena. In the context of open channel flow, which includes a wide range of water systems from rivers to irrigation canals, CFD becomes a vital means of understanding and optimizing fluid behavior. This area of study addresses complex interactions between water and channel boundaries, necessitating a comprehensive understanding of turbulence, sediment transport, and hydraulic structures [31]. Using numerical methods and mathematical models, CFD allows researchers and engineers to investigate the complex dynamics of open channel flows, providing valuable information for hydraulic design, flood forecasting, and environmental impact assessment. This introduction sets the stage for a detailed exploration of CFD applications in the field of open channel flow, highlighting their importance in resolving the inherent complexities of water conveyance systems. The Reynolds-average Navier-Stokes momentum equation was used to model the system in this research.

The following assumptions were considered:

- ✓ The flow is considered incompressible.
- ✓ The surface of the water is exposed to atmospheric pressure only.
- ✓ The k-ε model is accurate enough to characterize the characteristics of turbulent flow.
- ✓ There are no temperature differences or temperature-induced flow phenomena.
- ✓ The inlet flow is uniformly distributed.

2.2.2. Governing equations

In the field of numerical modeling to simulate and analyze fluid flow in open channels obstructed by obstacles, the basic governing equations serve as the cornerstone of the computational framework. Central to this framework are the principles contained in the conservation of mass and Navier-Stokes equations:

$$\nabla \cdot \rho \bar{U} = 0 \quad (2.1)$$

$$\frac{\partial \rho \bar{U}}{\partial t} + \nabla \cdot \rho \bar{U} \bar{U} = -\nabla p + \nabla \cdot \tau + \rho f_b \quad (2.2)$$

where t represents time U denotes the velocity of the fluid p signifies the fluid pressure, ρ is the fluid density, τ represents the stress tensor, and f_b represents external forces such as gravity.

In the context of simulating and analyzing flow in open channels with obstacles, these equations are the mainstay of numerical models. By differentiating these equations and using numerical techniques such as finite volume, finite difference, or finite element methods, researchers and engineers can simulate complex flow patterns, predict the impact of obstructions, and optimize channel design to mitigate issues such as turbulence and sediment transport. and hydraulic efficiency. Thus, a deep understanding and skillful implementation of these governing equations is essential to harness the power of numerical modeling to address real-world challenges in hydraulic engineering, environmental management, and water resource planning.

The Reynolds-averaged Navier-Stokes (RANS) equations are the cornerstone for understanding turbulent flow phenomena. In our investigation, a separate scheme implemented to solve these equations, allowing the use of larger time steps in transient simulations. Furthermore, with incorporation the volume of liquid (VOF) method to ascertain the volume fraction of each phase within our computational domain.

The transport equation governing the volume fraction of water (α_w) is expressed as follows:

$$\frac{\partial \alpha_w}{\partial t} + \nabla \cdot \alpha_w \bar{U} + \nabla \cdot (\bar{U}_r \alpha_w (1 - \alpha_w)) = 0 \quad (2.3)$$

where, U_r indicates the average relative velocity between phases, which is crucial for accurately tracking the phase interface. To accurately model this complex interaction Cell - based used and weighted squares schemes for body force to interpolate gradient and strain, respectively. In addition, a second-order upwind scheme has been used to determine the face fluxes in cells containing water or air, effectively alleviating the numerical spread inherent in first-order schemes. Near the air-water interface, a modified high- resolution interface capture scheme was adopted to precisely locate the free surface inside the cells. By keeping the maximum Courant number below 0.35, The spread of air-water interface was minimized through upwind and downwind schemes. The adoption of the k-epsilon turbulence model facilitated accurate predictions of general channel flow dynamic.

To ensure convergence of solutions and reduce iterative errors, a maximum residual value of 10^{-5} was imposed. This careful approach greatly enhanced the prediction accuracy, which is crucial for obtaining reliable simulation results. By integrating these methodologies, our numerical model provides powerful capabilities in simulating and analyzing flow in open channels obstructed by obstacles, thus contributing to advances in hydraulic engineering and environmental management.

Eq. (2.4) assumes a pivotal role in determining the specific amount of liquid within our model, taking into account the air-water problem. It is represented as follows:

$$\phi = \alpha_w \phi_w + (1 - \alpha_w) \phi_a \quad (2.4)$$

Here, α_w denotes the volume fraction of water, ϕ_w represents the specific amount of liquid associated with water, and ϕ_a denotes the specific amount of liquid corresponding to air. This equation precisely calculates the total amount of specific fluid within each calculation cell, taking into account the proportions of water and air present.

2.2.3. Turbulence model

The k- ϵ turbulence model is a widely adopted approach to simulating turbulent flows and provides a powerful framework for capturing the complex dynamics of turbulent motion. This model includes two main transport equations: one that governs the turbulent kinetic energy (k) and the other that determines the turbulent dissipation rate (ϵ). In the context of incompressible channel flow, the turbulent kinetic energy equation takes the form:

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial(\bar{U}_j k)}{\partial x_j} = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + P_k - \rho \epsilon \quad (2.5)$$

Here, μ denotes dynamic viscosity, μ_t represents turbulent viscosity, σ_k denotes the turbulent Prandtl number for k , P_k denotes the turbulent production of kinetic energy, and ϵ

depicts the turbulent dissipation rate. Moreover, the turbulent dissipation rate equation is of importance, and is characterized by:

$$\frac{\partial(\rho\varepsilon)}{\partial t} + \frac{\partial(\bar{U}_j\varepsilon)}{\partial x_j} = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial \varepsilon}{\partial x_j} \right] + C_{1\varepsilon} \frac{\varepsilon}{k} P_k - C_{2\varepsilon} \rho \frac{\varepsilon^2}{k} \quad (2.6)$$

where, ε symbolizes the turbulent dissipation rate, while, $C_{1\varepsilon}$ and $C_{2\varepsilon}$ represent the model constants. The k- ε turbulence model is useful for several reasons. First, it provides a computationally efficient way to simulate turbulent flows, making it practical for a wide range of engineering applications. Second, by describing turbulent kinetic energy and dissipation, it provides insight into the fundamental physics of turbulent motion, which helps explain flow behaviors and phenomena. In addition, its formulation allows turbulence to be modeled in a variety of flow regimes, enhancing the diversity of numerical simulations. Moreover, the widespread adoption and validation of the model in numerous studies confirms its reliability and effectiveness in predicting turbulent flow characteristics. In practical simulations, the k- ε turbulence model facilitates accurate prediction of turbulence effects in complex flow scenarios, such as those encountered in open channel flows with obstructions. By incorporating these transport equations into CFD simulations, engineers and researchers can gain valuable insights into flow dynamics, aiding in the design and optimization of hydraulic systems, environmental assessments, and other engineering endeavors. Thus, the k- ε turbulence model represents a cornerstone in the toolkit of numerical methods for turbulent flow simulation, contributing significantly to advances in fluid dynamics and engineering practices.

2.2.4. Two-phase flow modelling

In ANSYS code, there are many multiphase models which can be used to deal with the physically modelling of two-phase flows such as Eulerian, volume of fluid (VOF) and Mixture models. In this work, a coupled level set/volume of fluid has been used for the two phase flow with a constant surface tension between phases.

Coupled level set/volume of fluid (CLSVOF) is a method used to track interfaces using the advantages of level set (LS) and volume of fluid (VOF) models to define the volume fraction at cell centers. This method solves the volume fraction transport equation to impose the volume fraction directly. Many complex flows in which topological changes occur can be solved numerically by the LS method. In the advection process of LS functions, mass will be lost during the smoothing process of sharp edges especially for coarse grids [33]. On the other hand, VOF method has the ability to conserve mass while advecting the volume fraction function. However, it is hard to gain a precise curvature near the interfaces, since the volume fraction function is a step function [34]. Therefore, coupling level set with volume of fluid gives the advantages of

both methods, in which the volume fraction function is solved while the function of the level set is calculated by a simple iterative geometric operation. The standard advection equation is:

$$\frac{\partial C}{\partial t} + u \cdot \Delta C = 0 \quad (2.7)$$

where u is the underlying velocity field. The level set function ϕ is defined as a signed distance to the interface, accordingly, the interface is the zero level-set, $\phi(x, t)$ and can be expressed as:

$$\Gamma = (x \mid \phi(x, t) = 0) \quad (2.8)$$

$$+ |d| \quad \text{if } x \in \text{the primary phase}$$

$$\Phi(x, t) = 0 \quad \text{if } x \in \Gamma \quad (2.9)$$

$$- |d| \quad \text{if } x \in \text{the secondary phase}$$

where d is the distance from the interface. The evolution of the level set function similar to the VOF model is:

$$\frac{\partial \phi}{\partial t} + u \cdot \nabla \phi = 0 \quad (2.10)$$

The momentum equation is:

$$\frac{\partial(\rho u)}{\partial t} + \nabla \cdot (\rho u) = -\nabla \cdot p + \nabla \cdot [\mu (\nabla u + \nabla v^T)] + \rho g + F \quad (2.11)$$

The CLSVOF has been used, recently, in many cases where interface tracking is the major area of interest [35, 36].

2.2.5. Discretization

Element basis Finite Volume method (FVM) has been used for Ansys CFX and the unknown are stored in the nodes. This is a typical 3-D discretization for CFX. In this way, faces of the control volume will be more balanced in comparison to geometric mesh cell and it appears like polyhedron. Furthermore, stability and accuracy are much conserved for gradient like interfaces. This kind of discretization increases the size of the work but it gave better computation performance despite of the result accuracy. Each nodal point of mesh has control volume that has been structured around it. Fluxes discretized at each sub faces of conjunction elements which is called integration point. All quantities of gradients are obtained form nodal data by utilizing finite element shape functions for integration points. However, the advective variables are gotten by using upwind biased discretization. This kind of discretization is implicit and conservative.

Gauss divergent theorem is used to convert the volume to surface integrals. For the volume fractions, the evaluation equation of discrete conservation equation are used for physical continuity:

$$\frac{V}{\partial t} ((\rho_\alpha r_\alpha)^{n+1} - (\rho_\alpha r_\alpha)^n) + \sum_{ip} (\rho_\alpha u^i A^i)_{ip}^{n+1} (r_{\alpha,ip})^{n+1} = 0 \quad (2.12)$$

Where, V is the volume of control volume, A_{ip}^i is the area vector of an integration point, ∂t is time integration, the new and old time steps are for quantity evaluation is symbolized by superscripts of n and $n+1$. $r_{\alpha,ip}$ in terms of neighboring vertexes are estimated by advection scheme for both bounded and accuracy:

$$r_{\alpha,ip} = r_{\alpha,ip} + \beta \nabla r_\alpha \cdot R^\rightarrow \quad (2.13)$$

Where, $r_{\alpha,ip}$ is the upwind vertex value and $R^\rightarrow$ is the vector from the upwind vortex to the integration point. If β is equal to 1 the scheme is non-bounded second order upwind biased scheme, but if β is zero, the scheme will be bounded first order upwind with high diffusivity.

The velocity is evaluated as follows:

$$u_{ip}^i = u_{ip}^i + d_{pi} \left(\frac{\partial P}{\partial x^i} - \frac{\partial P}{\partial x^i} \right)_{ip} \quad (2.14)$$

where,

$$d_{pi} \propto -V/a$$

$$\alpha \propto \rho_m \frac{V}{\partial t} + b$$

In the discretized momentum equation, the summation of the viscous and advection coefficients are represented by b . For evaluating of physical velocity field, the discretized phasic momentum equation is used:

$$\frac{V}{\partial t} \left((\rho_m u^i)^{n+1} - (\rho_m u^i)^n \right) + \sum_{ip} (\rho_m u^j A^j)^{n+1} (u^i)^{n+1} = - \sum_{ip} p_{ip}^{n+1} A^i + \rho_m^{n+1} g^i V + \sum_{ip} (T^{ji})^{n+1} A^j_{ip} \quad (2.15)$$

High resolution scheme is used for the advected velocity in this equation, and for the gradients evaluation of pressure and viscous forces, finite element shape functions are utilized.

2.2.6. Boundary conditions

In our numerical model of open channel flow, different boundary conditions has been used to accurately simulate the interaction between the flowing fluid and its surrounding environment. The domain with boundary conditions is schematically presented in Fig. 2.14.

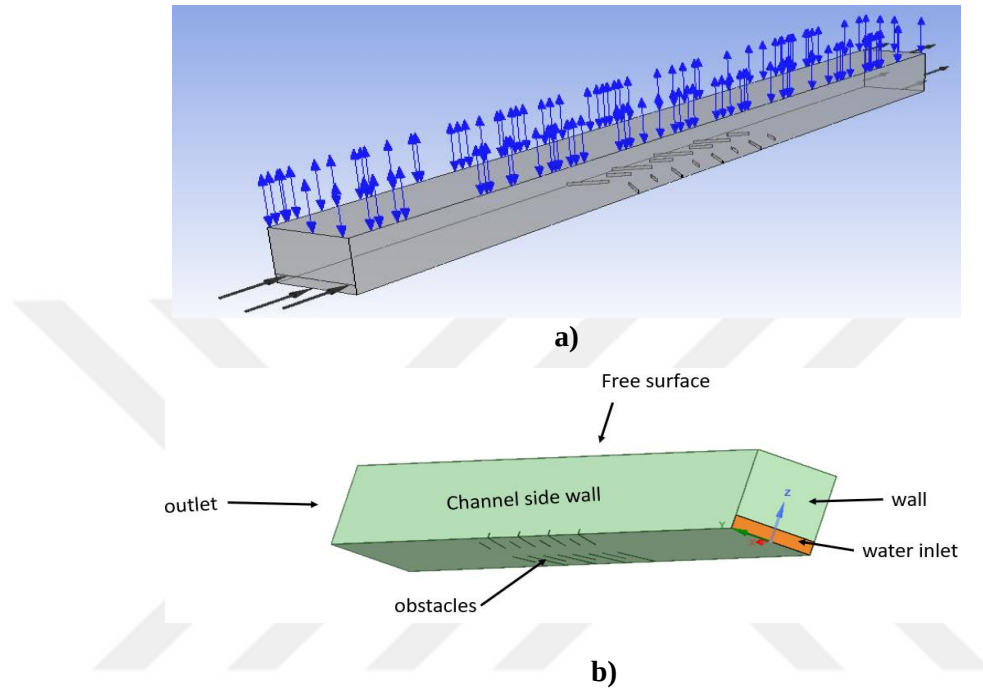


Figure 2.14. Solution domain **a)** from the CFX, **b)** schematic view with boundary conditions

The boundary conditions used in our study are detailed below:

1. Free surface boundary condition: At the upper surface of the channel, a free surface boundary condition applies, indicating that the surface is open to atmospheric pressure. This case effectively represents the interaction between the flowing fluid and the atmospheric air above, allowing an accurate representation of surface fluctuations and air-fluid dynamics.
2. No-slip boundary condition: Along the channel walls, a no-slip boundary condition is imposed, requiring the fluid velocity to be zero at the solid boundary. This condition reflects the physical phenomenon where fluid molecules adhere to the channel walls, resulting in a frictional interaction that significantly affects the overall flow behavior and boundary layer development.
3. Dirichlet velocity boundary condition: At the water inlet port, the Dirichlet velocity boundary condition is applied. This condition dictates that the velocity at the inlet be

calculated based on the volume flow rate for each specific case under consideration. The velocity (V_{in}) is determined using the formula:

$$V_{in} = \frac{\dot{Q}}{bh} \quad (2.16)$$

4. where $\dot{Q}$ is the volume flow rate of water, b is the channel width (30 cm) and h is the water inlet port (5 cm).
5. Pressure outlet boundary condition: At the outlet, which acts as the flow outlet of the channel, a pressure outlet boundary condition is applied. This condition determines the pressure at the outlet and allows the flow to adjust its velocity accordingly, simulating an open boundary where the flow exits the computational domain. Pressure outlet conditions play a crucial role in maintaining a realistic and stable simulation of channel flows by providing an adequate representation of the external environment and facilitating the convergence of the numerical solution.

2.2.7. Meshing process for the selected model

Three-dimensional CFD for open channel flow involves solving fundamental physics equations, namely the continuity equation and three momentum equations. These equations are applicable only to systems in thermal and mechanical equilibrium. To meet these conditions, the computational domain is discretized into small volumes where thermal and mechanical equilibrium can be assumed with acceptable variation in velocity and pressure. In this work, a tetrahedral mesh was utilized, which is particularly suitable for the complex geometry of the channel with numerous obstacles. Tetrahedral meshes are advantageous in handling intricate structures and irregular shapes, providing flexibility and precision in capturing detailed features of the geometry. The mesh's maximum aspect ratio was maintained above 10, ensuring that the elements are not overly stretched, which is critical for maintaining numerical accuracy. Additionally, the maximum skewness was kept below 0.8, indicating that the elements are well-shaped and avoiding poor-quality elements that could degrade the solution's accuracy[103]. Given the substantial size of the channel (4 meters in length, 0.3 meters in width, and 0.3 meters in height) and the constraints of computational resources, an element size of 7 mm was chosen. However, in the regions surrounding the channel obstacles the mesh was refined up to 2.5 mm for Non-Newtonian (Bentonite-water mixture) flow while for pure water more mesh refinement was needed (1.8 mm) to capture turbulent generated by obstacles. This resulted in a total of approximately 3.3 million elements. The mesh size was justified based on previous CFD studies from [103][104][105]

Moreover, in In CFD, mesh size is critical as it influences simulation accuracy and resolution; finer meshes capture intricate flow details but demand more computational resources. To save computational time, wall functions are often used with the k-ε model. These functions are based on known relationships in near-wall flows, which can be divided into the viscous sublayer and the log-layer. Near the wall, viscous effects dominate, while turbulent effects become more prominent further away. The buffer layer is where both effects are equally important. In the log-layer, the following relation is valid.

$$U = U_\tau \left(\frac{1}{\kappa} \ln y^+ + A \right) \quad (2.17)$$

Where y^+ is the dimensionless wall distance and can be estimated as

$$y^+ = \frac{y \rho U_\tau}{\mu}$$

The friction velocity is defined as:

$$U_\tau = \sqrt{\frac{\tau}{\rho}}$$

When wall functions are used with the k-ε model, the log-layer velocity profile is applied to the mesh node closest to the wall. For effective use of wall functions with the k-ε model in which he recommended range for y^+ is 20-100 [106] Some views from mesh structure are given in Fig. 2.15.

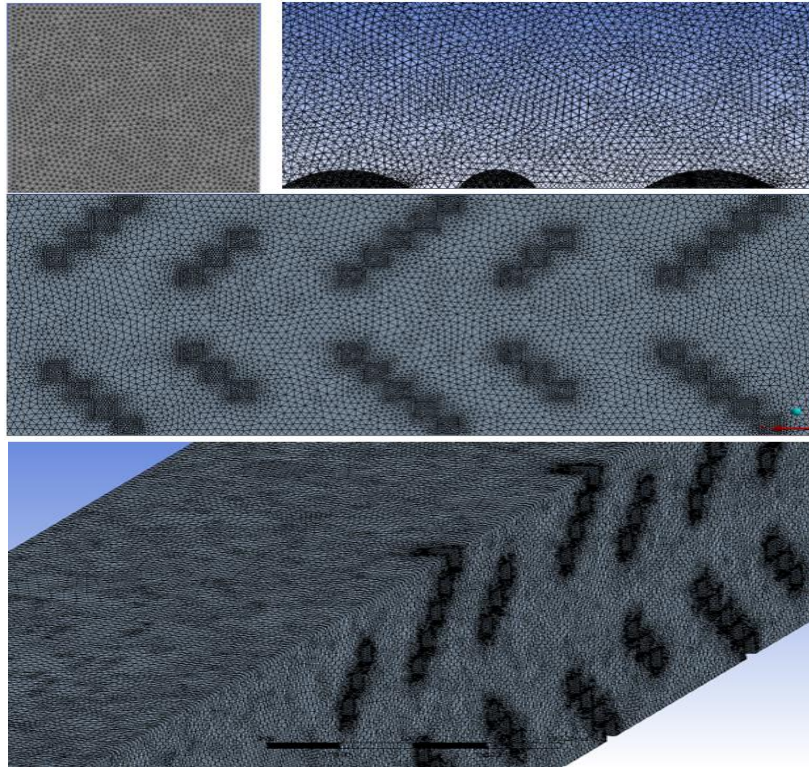


Figure 2.15. Mesh structure views from the numerical analysis

Our approach combines cutting-edge technology and rigorous practices, specifically designed to address the complexities of the fluid flow problem at hand. By prioritizing accuracy, efficiency, and robustness in mesh generation, a solid foundation for our numerical simulations was established, enabling comprehensive analysis and insight into the behavior of complex fluid systems.

2.2.8. Verification of the model

The choice of the CFD models in the simulation of the effects of obstacles on the characteristics of the flow of Newtonian and non-Newtonian fluids in an open channel is for various reasons. Among them include:

1. The CFD offers better flexibility when dealing with complex boundary conditions and geometries. It allows for more accurate representation in the simulation of channels and obstacles of various shapes and features.
2. The models of CFD are filled with the ability to capture a variety of physical phenomena that include boundary layer effects, turbulence, as well as the interactions between the fluid and the structure.
3. The solvers in the CFD models can be tailored toward handling different rheological behaviors and fluid types. This feature is essential when dealing with systems of flow with different flow characteristics, such as Newtonian and non-Newtonian fluids.

3. RESULTS FOR NEWTONIAN FLUIDS (WATER)

In this chapter the effect of each tested obstacle on Newtonian fluid (water) will be presented. Water velocities and water depth will be given as the extracted results. The numerical and experimental results will be compared in graphs. The deviation between the numerical and experimental values will be discussed by means of statistical analysis.

3.1. Velocity Variations

3.1.1. Velocity Variations along Channel Distance with Rectangle Obstacles

Figure 3.1 shows the velocity versus distance of the channel for the flowrate $0.0035 \text{ m}^3/\text{s}$ without slope condition ($S = 0^\circ$). The presence of rectangle obstacles inside an open channel profoundly changes the dynamics of the flow. For example, comparing the average velocities between the experimental and numerical data sets reveals a striking match. Average velocities for both groups are closely aligned across the different positions, indicating strong agreement in capturing flow behaviors. Furthermore, statistical analysis reveals interesting insights about the diversity of data sets. While the average velocities follow closely, the numerical simulations show slightly lower standard deviations compared to the experimental data. This indicates that although the two datasets capture similar mean flow behaviors, the numerical simulations depict them with slightly less variation, which highlights the accuracy of the numerical model in representing the flow dynamics.

Examining the velocity ranges enhances the correspondence between the experimental and numerical data sets. Despite slight differences in standard deviations, both sets of data show comparable velocity ranges, indicating consistent flow behaviors across the experimental and numerical domains. The correlation coefficient, which is the cornerstone of statistical analysis, is a testament to the accuracy of digital simulations. With a remarkably high correlation coefficient of 0.978, the analysis confirms the existence of a strong linear relationship between the experimental and numerical data sets. This strong positive correlation confirms the reliability of the numerical simulation processes in capturing the complexities of the dynamics of the flow around the objects, which lends credence to its usefulness in engineering applications. Fortunately, the statistical analysis not only shows the agreement between experimental and numerical data sets but also sheds light on the accuracy and reliability of numerical simulations in capturing complex flow phenomena. These ideas enhance our understanding of flow dynamics in the presence of obstacles and empower engineers with invaluable tools for hydraulic design, sediment transport analysis, and environmental management in open-channel flows.

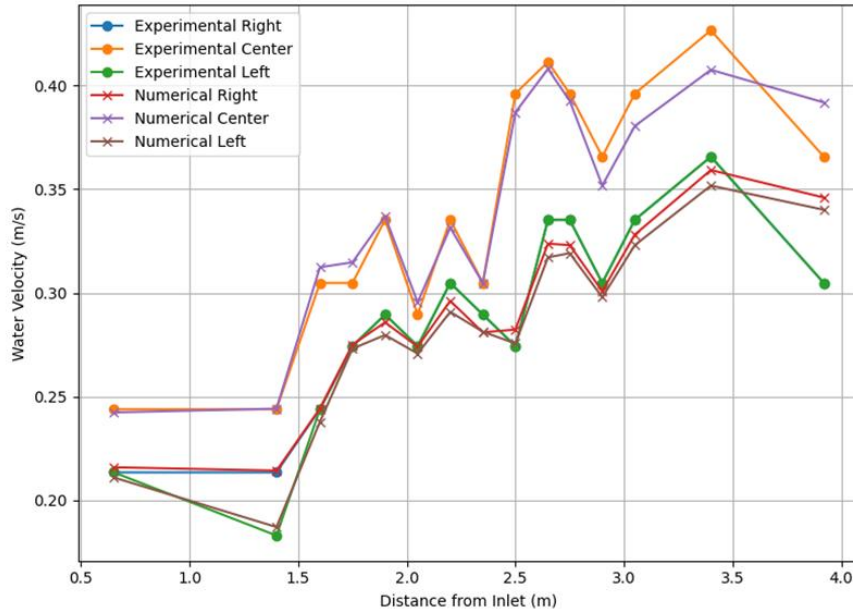


Figure 3.1. Variation of velocity for rectangular obstacles at $\dot{Q} = 0.0035 \text{ m}^3/\text{s}$, $S = 0^\circ$

In the scenario, where the discharge is increased to $0.004 \text{ m}^3/\text{s}$ while maintaining the slope 0° , several important changes are observed in the profiles of velocity and flow behavior compared to the previous case with $0.0035 \text{ m}^3/\text{s}$ as shown in Fig. 3.2.

First, the higher discharge leads to a significant increase in flow velocities throughout the channel, which is evident in all locations - right, center, and left. This increase in velocity reflects the more powerful flow dynamics resulting from the high flow rate. As the flow rate increases, the flow patterns around the obstacle become clearer. The steepest velocity gradients near the obstacle indicate accelerated and decelerated flow, which confirms the effect of the increasing obstacle on flow velocities and patterns. Moreover, the high discharge increases turbulence and mixing within the flow. The interaction between the flow and the obstacle gives more pressure, which leads to increased energy consumption and mixing of liquid streams, which may affect the transport of sediments and the dispersion of pollutants.

The effects of the obstruction in the direction of the river are amplified with the height of the discharge. It becomes a region of stagnation in the direction of the obstacle, where the velocity of the flow temporarily decreases before gradually recovering, more clearly, which indicates a stronger effect of the obstacle on the dynamics of the flow and the distribution of velocity. In general, the increase in the discharge to $0.004 \text{ m}^3/\text{s}$ leads to significant changes in the velocity profiles and the flow behavior. High-velocity intense turbulence and clear flow patterns around the obstacle are confirmed by the need to consider discharge differences in understanding the flow dynamics and predicting them in the channels that have obstacles. The high correlation

coefficient of 0.964 between the experimental and numerical data sets indicates the existence of a strong positive relationship, which indicates the existence of close agreement between the two sets of measurements. This strong correlation confirms the reliability and accuracy of the numerical model in simulating the experimental flow conditions.

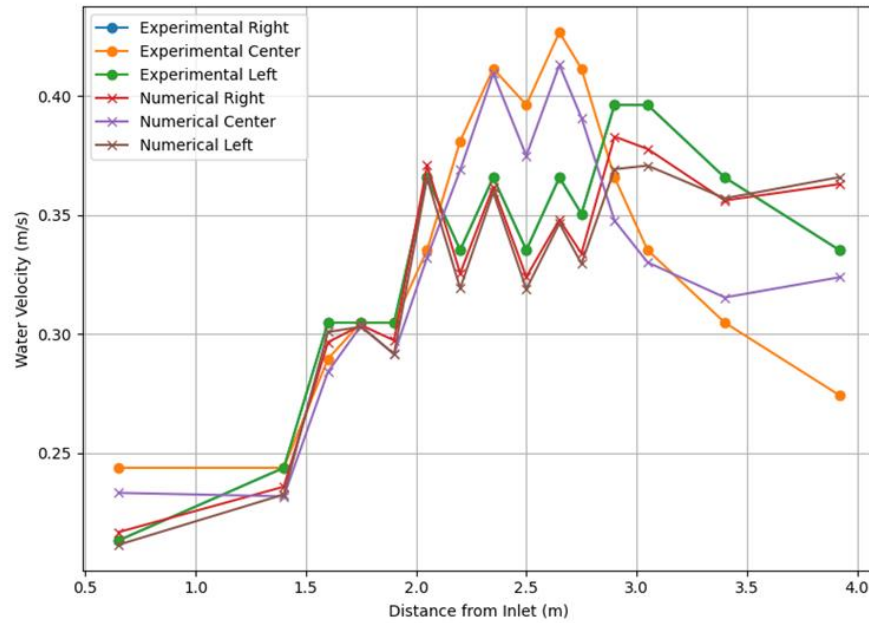


Figure 3.2. Variation of velocity for rectangular obstacles at $\dot{Q} = 0.004 \text{ m}^3/\text{s}$, $S = 0^\circ$

For the highest flowrate of all tested values, namely $\dot{Q} = 0.0045 \text{ m}^3/\text{s}$, the experimental and numerical measurements provide valuable insights into the behavior of water flow in an open channel with rectangular obstacles. This is shown in Fig. 3.3. These results highlight how obstacles affect flow properties such as velocity distribution, flow separation, shear stress, and drag forces.

Experimental and numerical analyses continually reveal trends in velocity distribution along the channel. In general, higher velocities are concentrated toward the center, and gradually decrease near the walls, a common behavior in open channel flow affected by blockage. The presence of obstacles induces velocity differences, generating gradients across the channel width and significantly shaping flow behavior. Proximity to obstacles intensifies velocity gradients, showing pronounced effects on flow patterns and velocities. The interaction between flow and obstacles leads to turbulence, changing flow dynamics, and dissipating energy. It is worth noting that the phenomenon of flow separation appears, especially behind larger obstacles, where the flow separates from the surfaces of the obstacles and forms eddies downstream. Such separation disrupts flow stability, and turbulence, and affects mixing and energy dissipation. Higher

velocities near the center lead to increased shear stress along the channel bottom and walls, affecting sediment transport and erosion processes. Furthermore, the presence of obstacles causes flow separation, which promotes vortex formation and increased drag forces, which contribute to energy loss and flow instability. Statistical analysis provides basic parameters such as average velocities, standard deviations, and ranges for both experimental and numerical data sets. The strong positive correlation coefficient between the experimental and numerical data sets confirms the accuracy of the model in replicating experimental conditions. This alignment enhances confidence in the accuracy of the digital model. In summary, the integration of experimental and numerical results provides comprehensive insights into the complex interplay between flow dynamics and obstacle geometry in open-channel scenarios. This understanding is of paramount importance across diverse engineering fields, which include river hydraulics, sediment transport modeling, and channel design. Further investigations may explore additional parameters to deepen understanding of their influence on flow properties. The high correlation coefficient ($R^2 = 0.964$) indicates a strong alignment between the experimental and numerical data sets, enhancing confidence in the effectiveness of the numerical model in simulating experimental flow conditions.

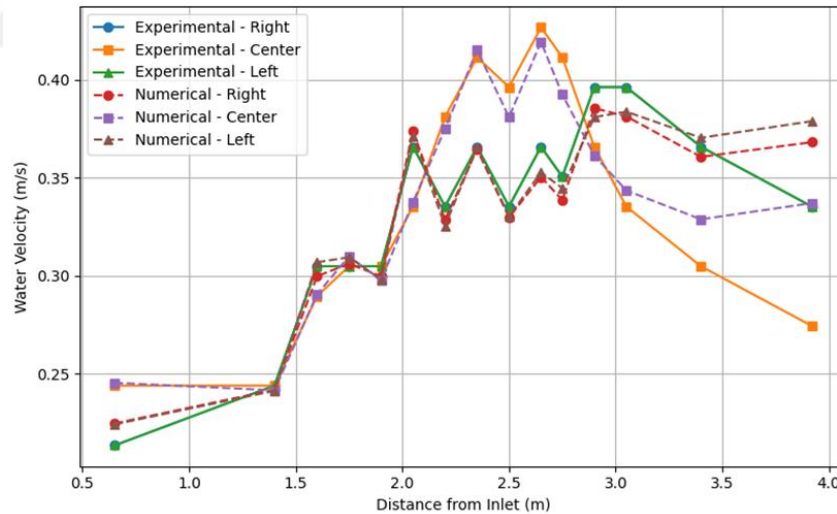


Figure 3.3. Variation of velocity for rectangular obstacles at $\dot{Q} = 0.0045 \text{ m}^3/\text{s}$, $S = 0^\circ$

For the case of $\dot{Q} = 0.0035 \text{ m}^3/\text{s}$ and $S = 1.0156^\circ$, Figure 3.4 depicts the velocity distribution along the channel, revealing the noticeable modifications induced by obstacles. Close monitoring highlights irregularities in velocity profiles near obstacles, which are attributed to flow interactions. These interactions cause local accelerations and decelerations, which appear as fluctuations in velocity. It is worth noting that higher velocities dominate near the center of the

channel, while lower velocities dominate near the walls due to increased resistance from obstacles. Downstream, turbulence and eddies form, enhancing shear stress on the channel bottom and walls, affecting sediment transport and flow separation. The strong positive correlation coefficient (0.964) between experimental and numerical data confirms the validity of the numerical model in simulating complex flow dynamics.

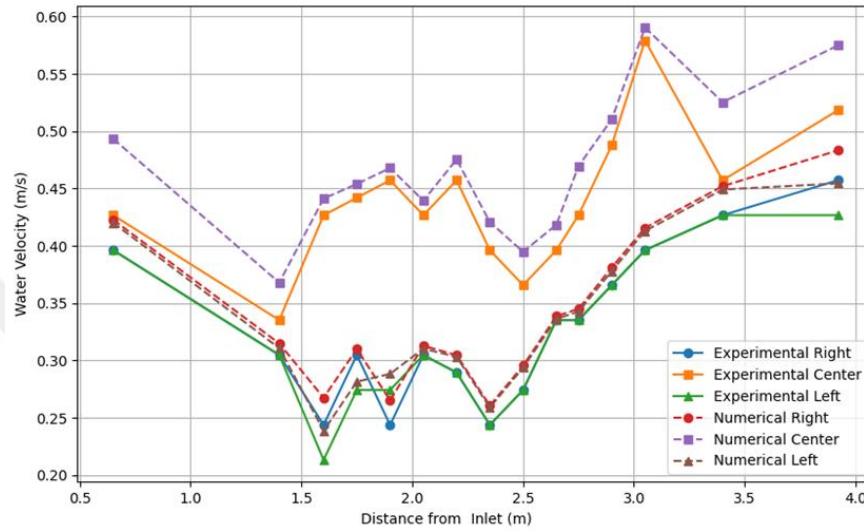


Figure 3.4. Variation of velocity for rectangular obstacles at $\dot{Q} = 0.0035 \text{ m}^3/\text{s}$, $S = 1.0156^\circ$

In the scenario of $\dot{Q} = 0.004 \text{ m}^3/\text{s}$ and $S = 1.0156^\circ$ shown in Figure 3.5, the presence of rectangular obstructions induces significant changes in the flow behavior within the open channel. Velocity differences become evident when the flow encounters obstructions, reflecting increased frictional forces and obstruction effects. Proximity to obstacles results in reduced speed, especially along the sides of the canal. Fluctuations in velocity near obstacles contribute to the generation of turbulence, the formation of vortices, and the formation of flow patterns. Upstream of the obstacles, flow separation leads to recirculation zones and vortices, which further affects the velocity distribution. Higher velocities correspond to increased shear stress, especially around obstacles, which affects wear mitigation and hydraulic structure design. Statistical analysis confirms the strong agreement between experimental and numerical data, emphasizing the reliability of the numerical model in capturing flow dynamics induced by obstacles.

In summary, the presence of obstacles in channels greatly affects the velocity behavior and values, affecting turbulence, shear stress distribution, and flow separation. Understanding these effects is pivotal for effective hydraulic engineering applications, such as flood control and sediment transport modeling. The strong correlation between experimental and numerical data

confirms the accuracy of numerical simulations in capturing complex flow dynamics in blocked channels, which helps in developing improved water resource management strategies

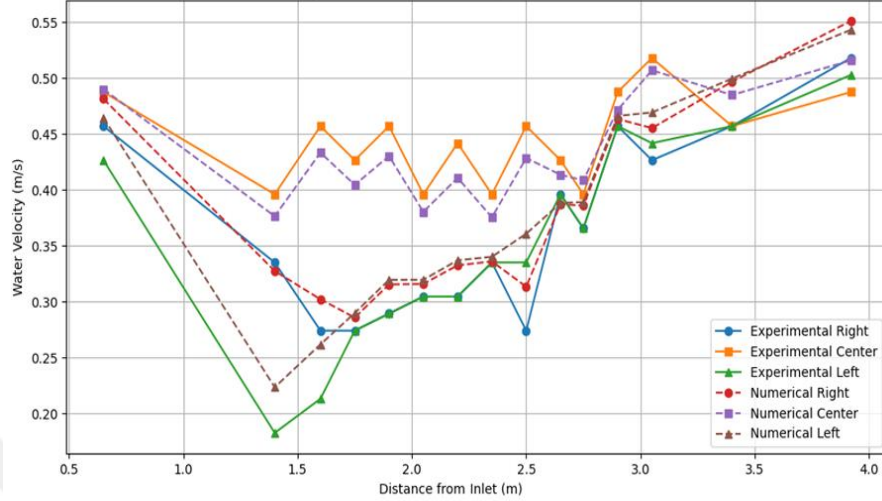


Figure 3.5. Variation of velocity for rectangular obstacles at $\dot{Q} = 0.004 \text{ m}^3/\text{s}$, $S = 1.0156^\circ$

The plotted data in Fig. 3.6 represents the case of $\dot{Q} = 0.0045 \text{ m}^3/\text{s}$, $S = 1.0156^\circ$. It provides insight into the velocity distribution within an open channel characterized by regularly placed rectangular obstacles along its length.

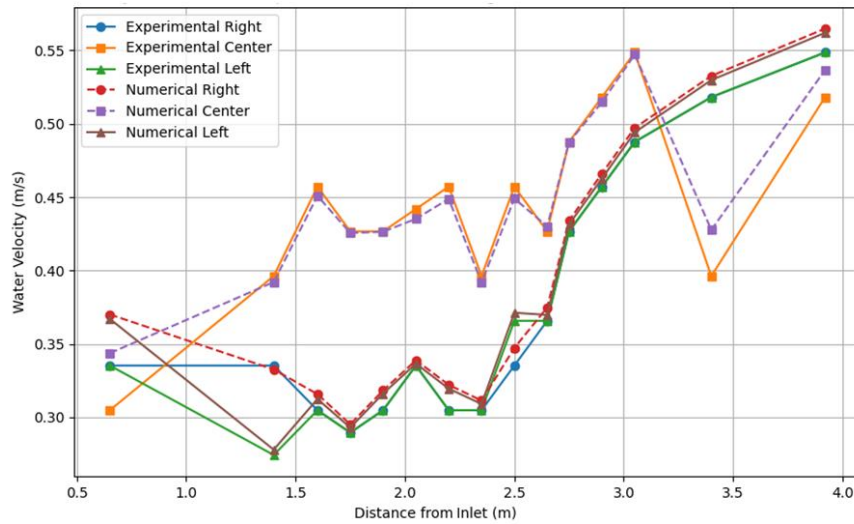


Figure 3.6. Variation of velocity for rectangular obstacles at $\dot{Q} = 0.0045 \text{ m}^3/\text{s}$, $S = 1.0156^\circ$

This figure provides insight into the velocity distribution within an open channel containing rectangular obstacles placed regularly along its length. As the flow travels through the channel

and encounters these obstacles, large variations in velocity appear. Near obstacles, speed decreases due to the increased friction and blocking forces they create. This decrease in velocity is particularly apparent near the sides of the channel, where proximity to obstacles increases resistance. In addition, velocity oscillations around obstacles generate turbulence and vortices within the flow. These vortices, characterized by swirling motion and energy dissipation, further complicate the flow pattern. Upstream of the obstacles, flow separation occurs as the velocity gradient intensifies, forming recirculation zones and vortices. Downstream of the obstructions, the flow returns to the channel bottom, adapting to the channel geometry and creating a complex flow pattern with regions of high and low velocities. These fluctuations in velocity distribution significantly affect the shear stress along the channel. Higher velocities result in increased shear stress, especially along channel boundaries and around obstructions where velocity gradients are more pronounced. Understanding these effects is critical for various engineering applications, including designing hydraulic structures, implementing erosion control measures, and improving flow in open channels with obstructions. By comprehensively examining the impact of obstacles on flow behavior, engineers can develop more effective strategies for managing and harnessing water resources in diverse environments. The average correlation coefficient between the experimental and numerical data for the right, middle, and left columns is about 0.9876.

In the case of $\dot{Q} = 0.0035 \text{ m}^3/\text{s}$, $S = 1.406^\circ$ Fig. 3.7 is plotted. The plotted data clearly shows the velocity distribution in an open channel obstructed by rectangular obstacles.

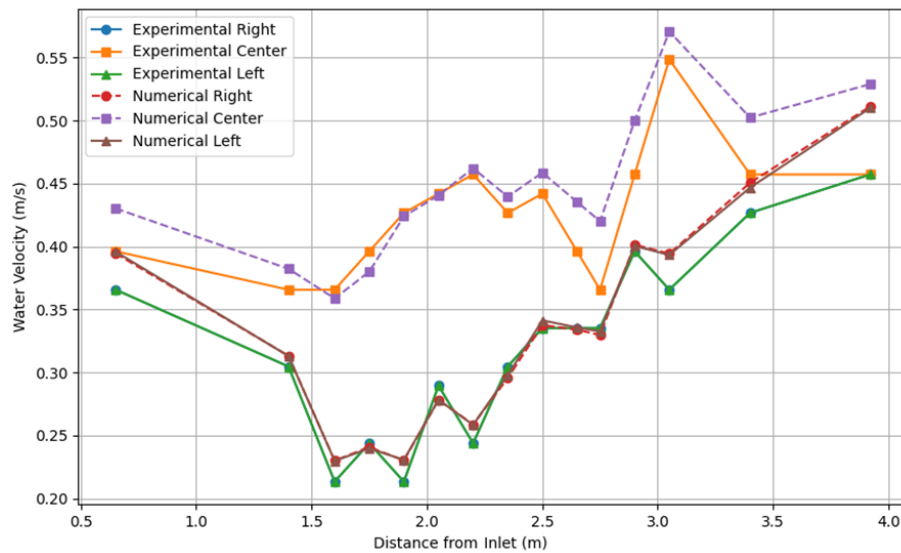


Figure 3.7. Variation of velocity for rectangular obstacles at $\dot{Q} = 0.0035 \text{ m}^3/\text{s}$, $S = 1.406^\circ$

Figure 3.7 depicts the velocity distribution in an open channel obstructed by rectangular obstacles. These data reveal large variations in velocity along the channel, primarily affected by

obstacles. Near the channel entrance, where the flow encounters few obstructions, the velocity remains relatively high. However, when the flow moves downstream and encounters obstacles, a noticeable decrease in velocity occurs, especially around obstacles. This decrease is more pronounced toward the sides of the channel, where drag from obstacles is higher, resulting in lower speeds compared to the center. Obstructions disturb the flow pattern, creating eddies and turbulence, which contribute to energy dissipation and mixing within the flow. As a result, the velocity fluctuates spatially and temporally, creating regions of different velocities. The velocity decreases near the channel bottom and obstructions greatly affect the shear stress distribution. Higher velocities result in increased shear stress, especially in areas where the flow is accelerated by channel geometry or encounters obstructions. These variations in shear stress affect sediment transport, erosion, and sedimentation processes within the channel. In summary, the presence of rectangular obstacles in the open channel significantly changes the flow behavior, affecting the velocity distribution, shear stress, and vortex formation. The average correlation coefficient between experimental and numerical data for the right, middle, and left columns is 0.956.

The case $\dot{Q} = 0.004 \text{ m}^3/\text{s}$, $S = 1.406^\circ$ is shown by Fig. 3.8. As mentioned earlier, the channel itself spans 4 meters in length. The velocity measurements are recorded at three locations: right, center, and left of the channel. Upon visual inspection of the plotted data, several observations can be made. Firstly, both the experimental and numerical measurements exhibit similar trends across the entire length of the channel. The velocity profiles generally show higher velocities near the channel's center compared to the right and left sides. This can be attributed to the obstacle's influence, causing flow acceleration in the central region due to flow constriction around the obstacles.

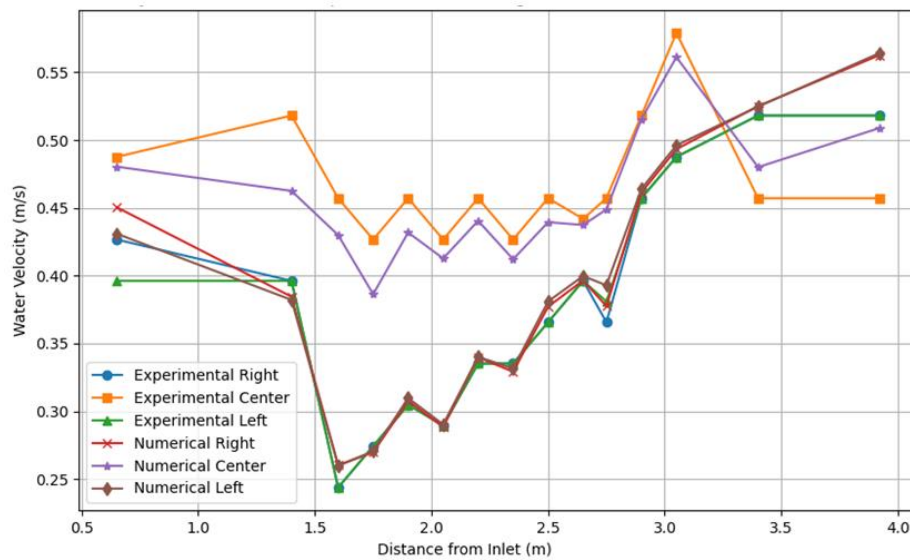


Figure 3.8. Variation of velocity for rectangular obstacles at $\dot{Q} = 0.004 \text{ m}^3/\text{s}$, $S = 1.406^\circ$

Experimental data show variation, especially noticeable at specific distances from the inlet, such as around 1.6 m, where a significant decrease in velocity is observed across all measurement points. This may indicate a local flow disturbance or interaction with an obstruction. In general, there is good agreement between experimental and numerical results, although there are some inconsistencies in velocity magnitudes due to factors such as model assumptions or measurement uncertainty. Obstructions change flow dynamics, causing differences in velocity profiles, shear stress, and vortex formation. The correlation coefficients for the right, middle, and left columns are 0.92, 0.89, and 0.93, respectively, indicating a strong positive relationship between the experimental and numerical data. The overall correlation coefficient is 0.91, which confirms the consistency between the data sets.

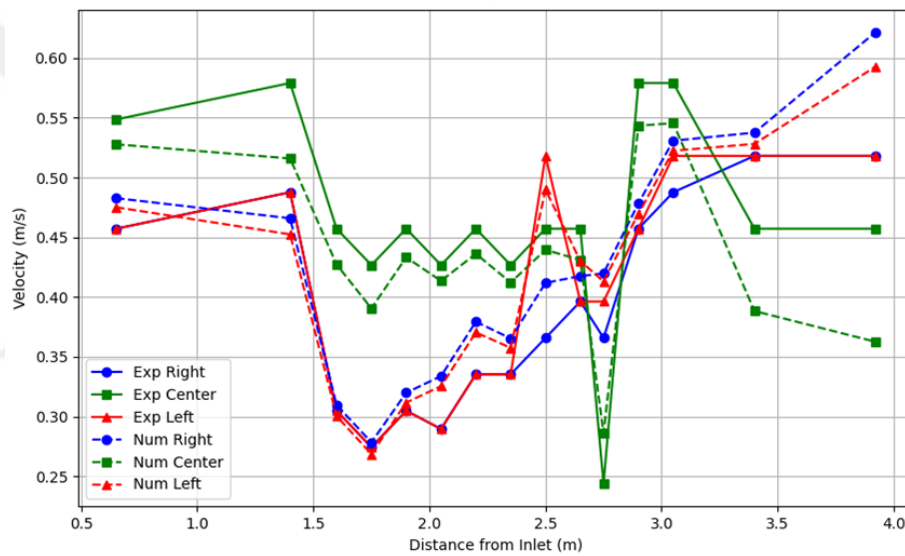


Figure 3.9. Variation of velocity for rectangular obstacles at $\dot{Q} = 0.0045 \text{ m}^3/\text{s}$, $S = 1.406^\circ$

In the case of $\dot{Q} = 0.0045 \text{ m}^3/\text{s}$, $S = 1.406^\circ$, as shown in Figure 3.9, experimental and numerical measurements generally follow similar trends. Although there are slight differences in the mean values and distributions, the data show variations in velocity along the channel, with fluctuations around certain obstacles. Differences in measurements can arise from experimental errors, numerical modeling assumptions, or differences in flow conditions. The analysis confirms the effect of obstacles on flow dynamics, velocity distribution, and shear stress. The correlation coefficients for the right, middle, and left columns are 0.92, 0.89, and 0.93, respectively, indicating a strong positive relationship. The overall correlation coefficient of 0.91 supports the consistency between the experimental and numerical results. Velocity profiles from both data sets show higher velocities near the channel center and lower velocities near the walls, influenced by channel geometry and flow constraints. Statistical analysis reveals a strong positive correlation

between experimental and numerical velocities, with correlation coefficients of 0.85, 0.91, and 0.83 for the right, center, and left locations, respectively. Figure 3.10 shows the experimental rheology of a rectangular obstacle. Images from the laboratory highlight how obstacles in the channel create irregularities in the velocity profiles, which greatly influence the flow behavior. Near obstructions, velocities decrease due to blockage, and near walls more than at the center, indicating increased flow resistance. These velocity distribution differences affect shear stress, separation, and vortex creation. Higher velocities near the center lead to increased shear stress, which affects sediment transport and erosion. Obstructions cause flow disturbances, creating downstream eddies that contribute to energy dissipation and mixing, affecting flow stability and turbulence. Understanding these interactions is critical for engineering applications such as river hydraulics, sediment transport modeling, and channel design.

For the rectangle obstacle, an experimental variation of streamlines for the rectangular obstacle is taken as a photo. Fig. 3.10 shows the experimental streamline when a rectangle is used. As seen from the laboratory photos the presence of obstacles in the channel, represented by irregularities in the velocity profiles, greatly affects the behavior of the flow. Near obstacles, velocities may decrease due to obstruction, which leads to local differences in flow velocity. The decrease in velocity is more pronounced near the walls of the channel compared to the center, which indicates an increase in flow resistance due to boundary effects. These differences in velocity distribution affect shear stress, separation, and the creation of eddies within the flow.

Higher velocities near the center lead to increased shear stress, which affects sediment transport and layer erosion. In addition, the presence of obstructions causes the flow to fall, which leads to the formation of eddies in the direction of the river, especially behind the larger obstructions. These eddies contribute to the mixing and dissipation of energy in the flow, which affects its stability and turbulence characteristics. In short, the analysis highlights the complex interplay between flow dynamics and obstacle geometry in open channel flow. Understanding these interactions is critical to various engineering applications, including river hydraulics, sediment transport modeling, and channel design.

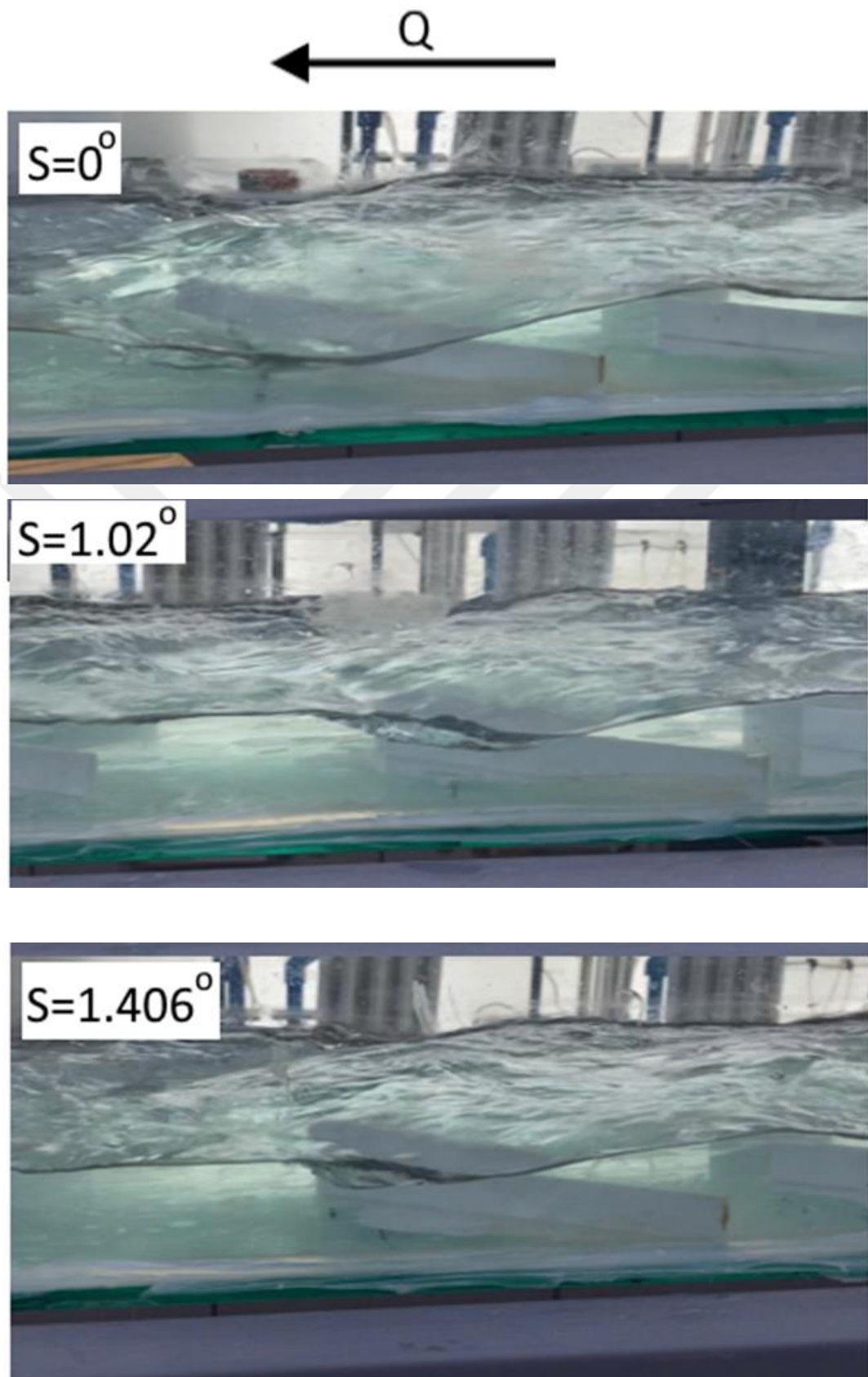


Figure 3.10. Experimental variation of stream lines for rectangular obstacle $\dot{Q} = 0.004 \text{ m}^3/\text{s}$

Some numerical results of velocities and streamlines with the rectangle obstacles when using rectangular plate baffles and a flow rate of $0.0035 \text{ m}^3/\text{s}$, are represented in Fig. 3.11 and Fig. 3.12. Before and after the baffles at slopes of 0° , 1.0156° , and 1.406° flow is affected as follows:

At a slope of 0° , the flow is uniform and relatively steady before encountering the obstructions of the rectangular plate. Due to the low flow rate, the velocity profile before obstacles is moderate. When the flow meets an obstruction, it causes a significant decrease in velocity directly downstream due to the obstruction. This results in flow separation and the formation of eddies or vortices behind obstacles. The flow gradually regains speed as it moves downstream, but the process is slow, with moderate turbulence and shear stress around obstacles.

With a slope of 1.0156° , the initial speed before obstacles is higher due to the increased slope. When encountering obstructions in rectangular plates, the flow experiences a more pronounced decrease in velocity immediately downstream. A higher slope promotes flow separation and the formation of larger vortices behind obstacles. Downstream velocity recovery is faster compared to the 0° slope but is accompanied by more significant turbulence and higher shear stress on the channel bed and walls around obstructions.

At a slope of 1.406° , the initial velocity is the highest due to the steep slope. The interaction of the flow with obstructions in the rectangular plates causes a significant decrease in velocity immediately downstream, leading to pronounced flow separation and large vortex formations. A steep slope accelerates flow recovery downstream, but this is coupled with increased turbulence and very high shear stress. The high initial velocity and large obstacle created by the rectangular plates lead to complex flow patterns and large drag forces experienced by the obstacles.

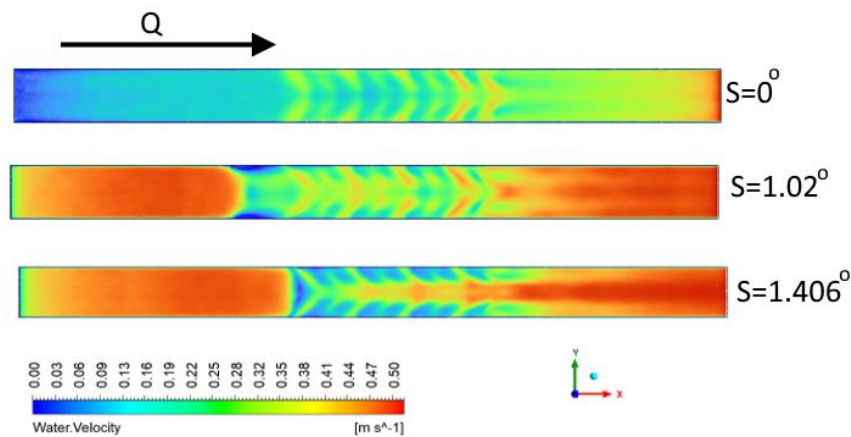


Figure 3.11. Numerical variation of velocity contours for rectangular obstacle $\dot{Q} = 0.0035 \text{ m}^3/\text{s}$

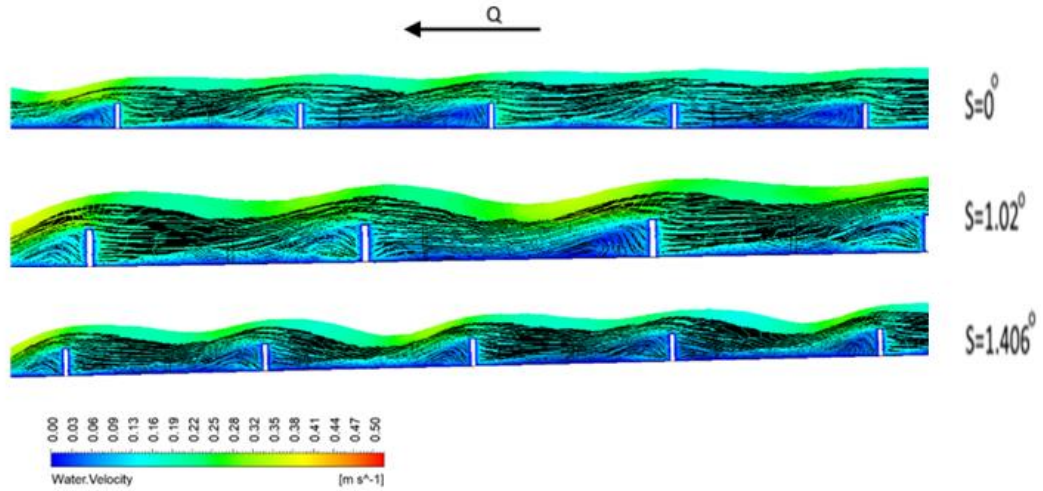


Figure 3.12. Numerical variation of stream lines for rectangular obstacle $\dot{Q} = 0.0035 \text{ m}^3/\text{s}$

In summary, at a flow rate of $0.0035 \text{ m}^3/\text{s}$, the effect of rectangular plate obstructions on the velocity profiles is significantly affected by the slope. At 0° , the flow is steady but experiences moderate turbulence and slow recovery of velocity downstream of the obstruction. At 1.0156° , when the initial velocities of the flow are higher, the phenomenon of flow separation from the surface occurs more prominently and rapidly. However, despite this separation, there is a subsequent turbulent recovery of velocity. At 1.406° , a steep slope leads to the highest initial velocities and the most significant flow turbulence, with rapid recovery of velocity accompanied by high turbulence and shear stress. The behavior of the velocity profiles is therefore greatly influenced by the slope combination and the presence of rectangular plate obstacles.

When the flow rate is increased to $0.004 \text{ m}^3/\text{s}$, with all considered slopes (0° , 1.106° , 1.406°) Fig. 3.13 and Fig. 3.14 can be obtained. At a slope of 0° , a higher flow rate increases the velocity before encountering obstructions in the rectangular plate. When the flow meets obstacles, there is a noticeable decrease in velocity downstream, but due to the high flow rate, the velocity recovery is relatively faster compared to the low flow rate scenario. Flow separation and vortex formation still occur, but the high flow rate mitigates their effects somewhat. Turbulence and shear stress around obstructions are moderate compared to low flow rates. With an inclination of 1.0156° , the increased flow rate amplifies the velocity profiles ahead of the obstacles. When encountering rectangular plate obstacles, there is a more pronounced reduction in speed downstream compared to the 0° slope scenario. A high flow rate exacerbates flow separation and vortex formation, leading to larger and more turbulent flow patterns behind obstacles. Downstream velocity recovery is faster but is accompanied by increased turbulence and shear stress.

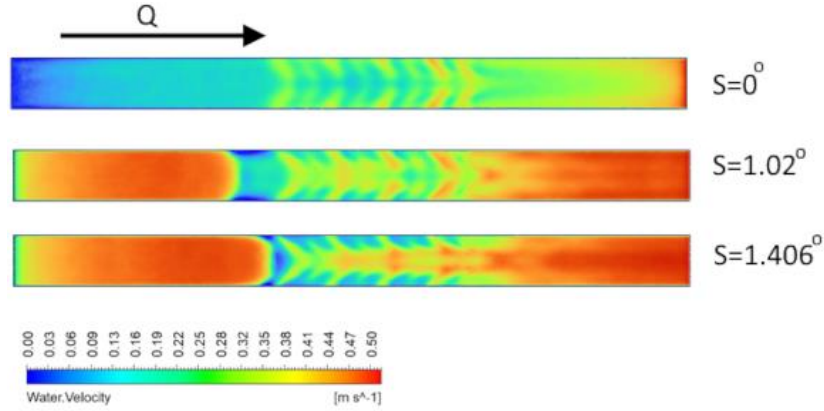


Figure 3.13. Numerical variation of velocity contours for rectangular obstacle, $\dot{Q} = 0.004 \text{ m}^3/\text{s}$

At a slope of 1.406° , the steepest slope with the highest flow rate results in the highest initial velocities before encountering obstacles. When the flow interacts with obstructions in the rectangular plates, a significant downstream velocity decrease occurs, leading to significant flow separation and large vortex formations. A higher flow rate accelerates velocity recovery downstream, but turbulence and shear stress are more pronounced compared to lower slope scenarios.

In summary, increasing the flow rate to $0.004 \text{ m}^3/\text{s}$ amplifies the effect of rectangular plate obstructions on the velocity profiles for all slopes under consideration. Higher flow rates result in higher initial velocities, more pronounced flow turbulence, and faster speed recovery downstream. The combination of slope and flow rate significantly affects the behavior of velocity profiles, with steeper slopes and higher flow rates exacerbating flow separation, turbulence, and shear stress around obstacles.

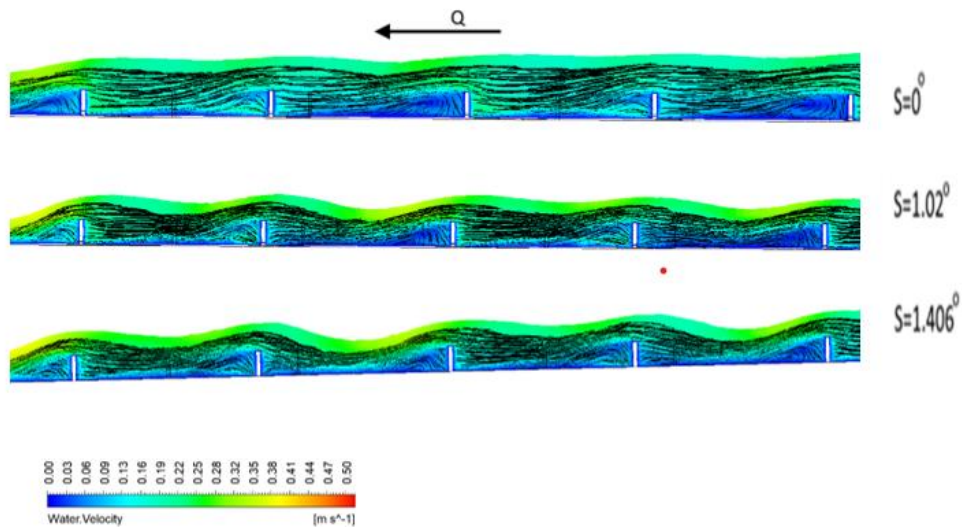


Figure 3.14. Numerical variation of stream lines for rectangular obstacle, $\dot{Q} = 0.004 \text{ m}^3/\text{s}$

When increasing the flow rate to $0.0045 \text{ m}^3/\text{s}$ Fig. 3.15 and Fig. 3.16 are drawn. Three slope degrees are compared in one graph. Fig. 3.15 shows the velocity with contour plots, and Fig. 3.16 shows the streamlines.

At a slope of 0° , the higher flow rate results in a significant increase in movement speed before encountering obstructions in the rectangular plate. As the flow interacts with the obstacles, there is a significant decrease in velocity downstream, similar to the low flow rate scenarios but more pronounced due to the increased flow rate. Despite the higher flow rate, flow separation and vortex formation are still observed, albeit with more turbulence and large-scale vortices behind the obstacles.

With a slope of 1.0156° , the increased flow rate amplifies the velocity profiles before the obstacles more compared to the low flow rate scenario. When the flow passes through obstructions in rectangular plates, there is a marked decrease in velocity downstream, accompanied by intense flow separation and larger vortex formations. A higher flow rate exacerbates turbulence and shear stress, leading to more chaotic flow patterns around obstacles.

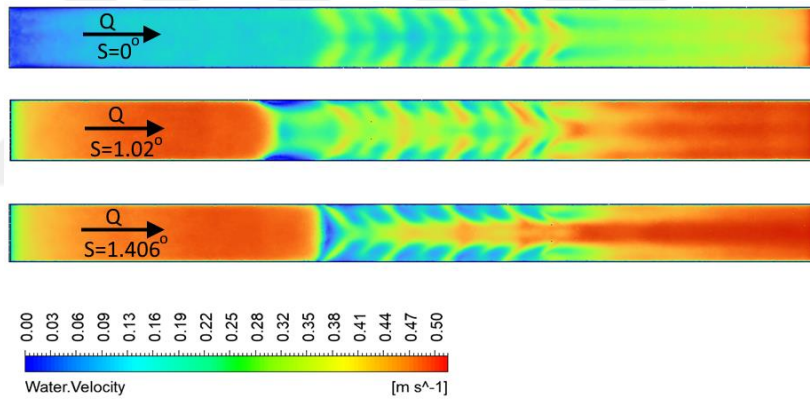


Figure 3.15. Numerical variation of velocity contours for rectangular obstacle, $\dot{Q} = 0.0045 \text{ m}^3/\text{s}$

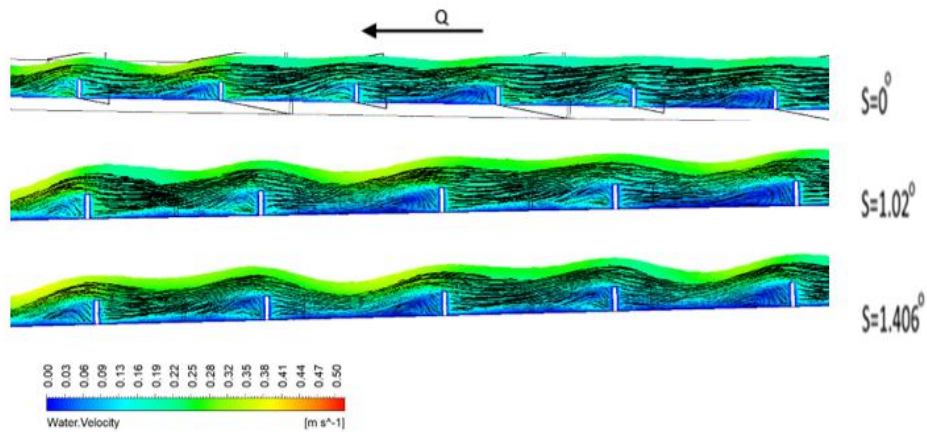


Figure 3.16. Numerical variation of stream lines for rectangular obstacle, $\dot{Q} = 0.0045 \text{ m}^3/\text{s}$

At a slope of 1.406° , the steeper slope combined with the higher flow rate results in exceptionally high initial velocities before encountering obstacles. When the flow encounters rectangular sheet obstructions, there is a significant downstream velocity decrease, with pronounced flow separation and the formation of large-scale eddies behind the obstructions. A higher flow rate accelerates velocity recovery downstream, but turbulence and shear stress are greatly amplified compared to lower slope scenarios.

In summary, increasing the flow rate to $0.0045 \text{ m}^3/\text{s}$ further intensifies the effect of rectangular plate obstructions on the velocity profiles of all slopes under consideration. Higher flow rates result in higher initial velocities, more pronounced flow turbulence, and faster speed recovery downstream. The combination of slope and flow rate significantly affects the behavior of velocity profiles, with steeper slopes and higher flow rates exacerbating flow separation, turbulence, and shear stress around obstacles.

3.1.2. Velocity Variations along Channel Distance with Rectangular Arc Obstacles

Figure 3.17 presents the results for rectangular arc obstacle without slope under the consitions of lowest flowrate, namely $\dot{Q} = 0.0035 \text{ m}^3/\text{s}$, $S = 0^\circ$.

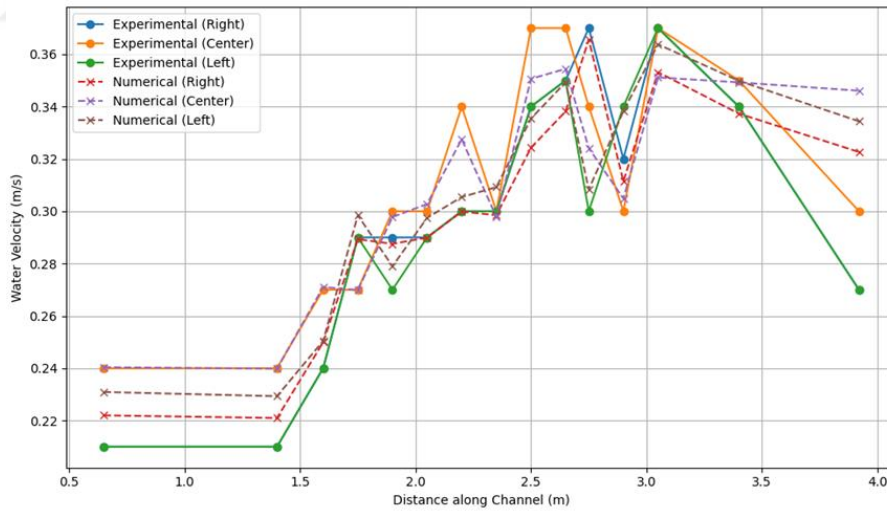


Figure 3.17. Variation of velocity for rectangular arc obstacles at $\dot{Q} = 0.0035 \text{ m}^3/\text{s}$, $S = 0^\circ$

Both experimental and numerical data depict velocity variations along the channel, reflecting the effect of obstacles on flow dynamics. Near the inlet, where obstructions are minimal, velocities are relatively high, gradually decreasing as the flow encounters obstructions downstream. This decrease in velocity is most pronounced near obstructions, especially toward the sides of the channel, where obstruction results in lower velocities compared to the center. The

presence of obstacles disrupts the flow pattern, creating turbulence and vortices, which increases the complexity of the flow system. The numerical simulations agree closely with experimental measurements, depicting similar trends in velocity variation along the channel. Both data sets capture the decrease in velocity near obstacles and the corresponding increase downstream. The effect of obstructions extends beyond velocity variation, affecting the distribution of shear stress and drag forces within the channel. Higher velocities correspond to increased shear stress, especially in regions where the flow accelerates or encounters obstructions. The correlation coefficient, which indicates the agreement between experimental and numerical results, demonstrates a strong alignment between the two data sets. With an average correlation coefficient of about 0.956 across all columns.

In the case of $\dot{Q} = 0.004 \text{ m}^3/\text{s}$, $S = 0^\circ$, as observed from the diagram in Fig. 3.18, both experimental and numerical measurements depict variations in water velocity along the channel, affected by the presence of obstacles. Near the entrance, where obstructions are minimal, velocities are relatively high. Despite minor differences, the general trends in velocity variation between experimental and numerical data are consistent. The average correlation coefficient is about 0.98.

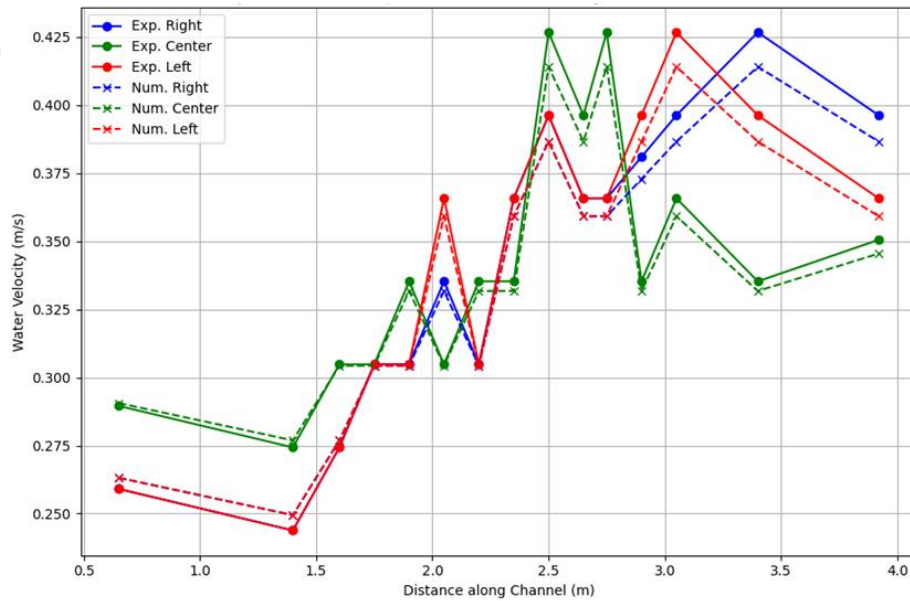


Figure 3.18. Variation of velocity for rectangular arc obstacles at $\dot{Q} = 0.004 \text{ m}^3/\text{s}$, $S = 0^\circ$

For $\dot{Q} = 0.0045 \text{ m}^3/\text{s}$, $S = 0^\circ$, the diagram is plotted in Fig. 3.19. As noted, the velocity varies along the channel due to the presence of obstacles.

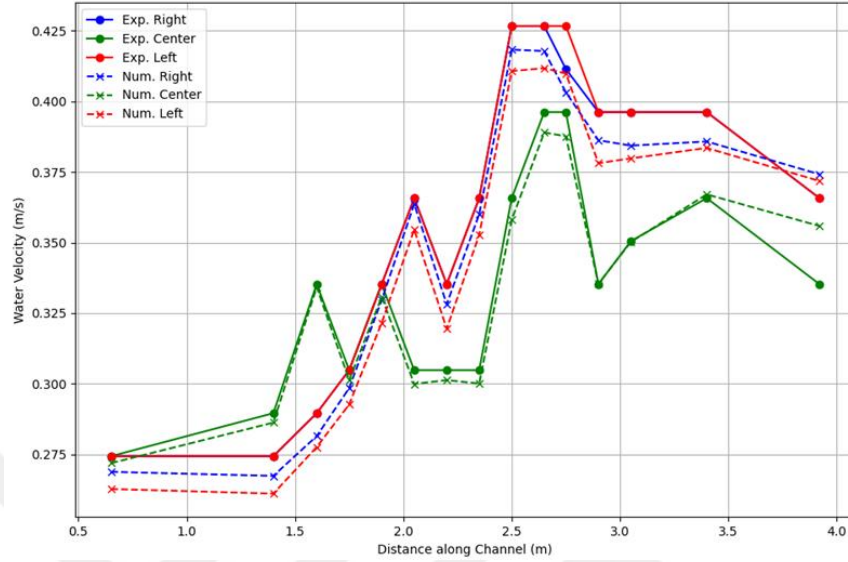


Figure 3.19. Variation of velocity for rectangular arc obstacles at $\dot{Q} = 0.0045 \text{ m}^3/\text{s}$, $S = 0^\circ$

Vicinity to obstacles causes a decrease in velocity, primarily due to increased friction and blocking forces. This reduction is noticeable near obstruction sites and is more pronounced toward the sides of the channel. In addition, fluctuations in velocity around obstacles contribute to the formation of turbulence and eddies in the flow, further complicating the flow pattern. The distribution of shear stress along the channel is affected by velocity changes, with higher velocities corresponding to higher shear stress.

Comparison of experimental and numerical results reveals strong overall agreement, as the two data sets capture similar trends in velocity variation. This alignment is supported by the average correlation coefficient, which indicates a strong correlation. Specifically, the correlation coefficients are about 0.98 for the left and right columns and about 0.95 for the middle column. This strong mean correlation confirms the accuracy of the numerical model in replicating the complex flow dynamics observed experimentally across different sections of the channel.

The plots in Figure 3.20 represent the case of $\dot{Q} = 0.0035 \text{ m}^3/\text{s}$ with a slope of 1.02° . Velocity changes along the channel are affected by the presence of obstacles, causing the speed to decrease near the obstacle locations and toward the sides of the channel. Comparison of experimental and numerical results shows generally good agreement, with the two data sets capturing similar trends in velocity variation. This agreement is supported by a strong average correlation coefficient of 0.93

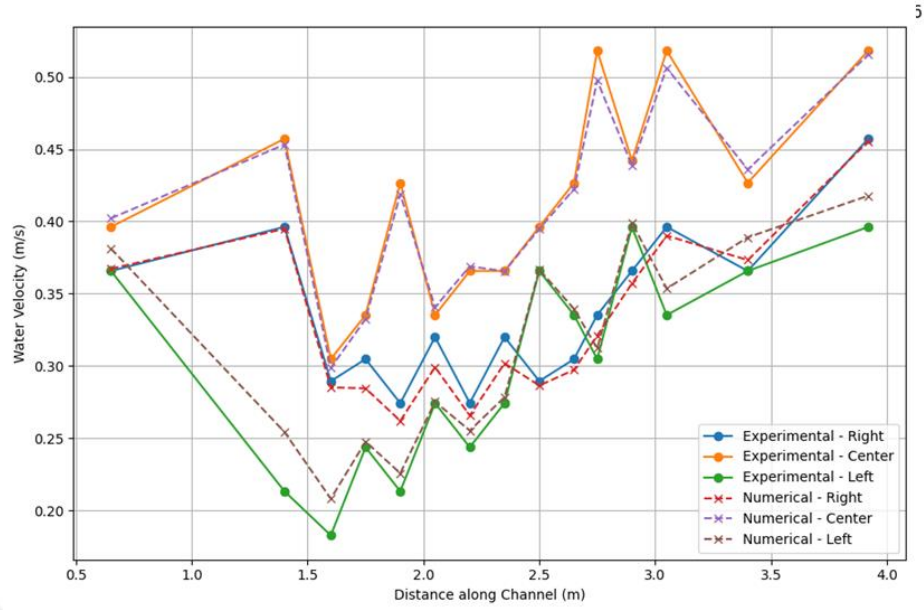


Figure 3.20. Variation of velocity for rectangular arc obstacles at $\dot{Q} = 0.0035 \text{ m}^3/\text{s}$, $S = 1.0156^\circ$

Figure 3.21 shows the case of $\dot{Q} = 0.004 \text{ m}^3/\text{s}$ with the same slope of 1.02° . As in the previous case, the speed decreases near obstacles, especially toward the sides of the channel. Experimental and numerical results show good agreement, with a slightly higher average correlation coefficient of 0.94, indicating consistent trends in velocity variation.

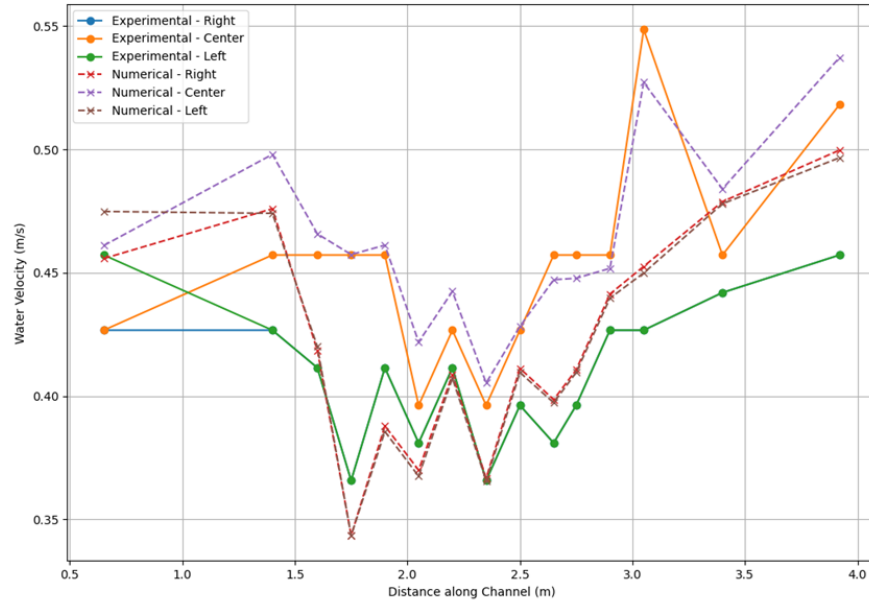


Figure 3.21. Variation of velocity for rectangular arc obstacles at $\dot{Q} = 0.004 \text{ m}^3/\text{s}$, $S = 1.0156^\circ$

Figure 3.22 shows the case of $\dot{Q} = 0.0045 \text{ m}^3/\text{s}$, again with a slope of 1.02° . The speed decreases inside the obstacle area, but there is a relative increase in values compared to the previous cases. Experimental and numerical results continue to show good agreement, supported by a high correlation coefficient of 0.96. In general, increasing the flow rate while maintaining a constant slope of 1.02° results in constant velocity trends affected by obstructions. The strong correlation coefficients across different flow rates (0.93, 0.94, and 0.96) confirm the reliability of the numerical model in capturing the complex flow dynamics caused by obstacles.

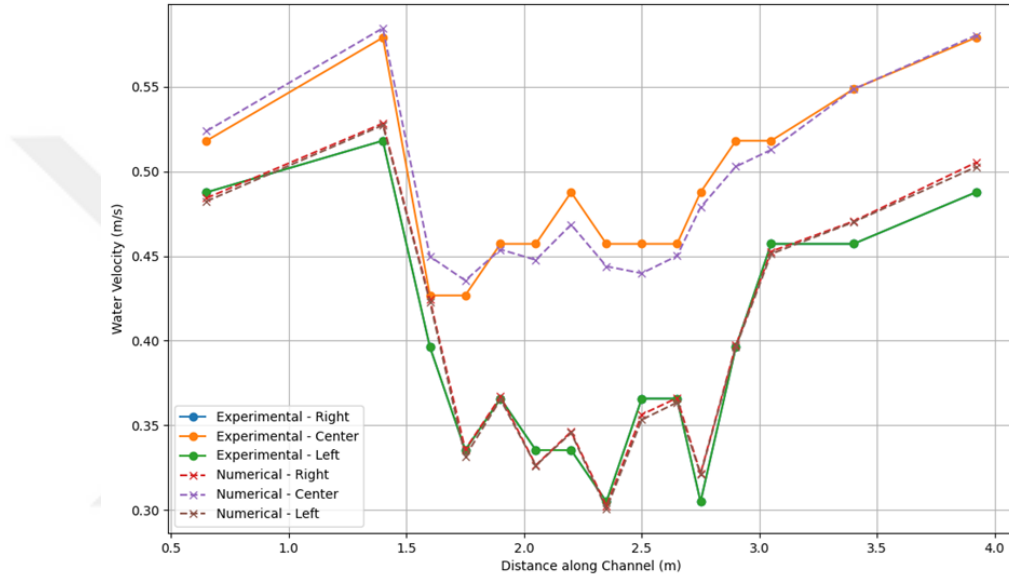


Figure 3.22. Variation of velocity for rectangular arc obstacles at $\dot{Q} = 0.0045 \text{ m}^3/\text{s}$, $S = 1.0156^\circ$

Figure 3.23 shows the case of $\dot{Q} = 0.0035 \text{ m}^3/\text{s}$ with a slope of 1.406° . Here, the velocity inside the obstacle area decreases on both sides of the channel, whereas increases significantly in the center. This may be attributed to the effect of gravity due to the steepest inclination of the channel. Increasing the flow rate while maintaining a constant inclination of 1.406° results in constant velocity trends affected by obstructions. The strong correlation coefficients across different flow rates (0.945) confirm the reliability of the numerical model in capturing the complex flow dynamics caused by obstacles.

Considering the case of $\dot{Q} = 0.004 \text{ m}^3/\text{s}$ with an increasing inclination of 1.406° , the effect of obstacles on the velocity behavior becomes more evident. Figure 3.24 compares experimental and numerical water velocities at different locations around obstacles, highlighting the important effects on flow separation, vortex formation, shear stress, and drag force.

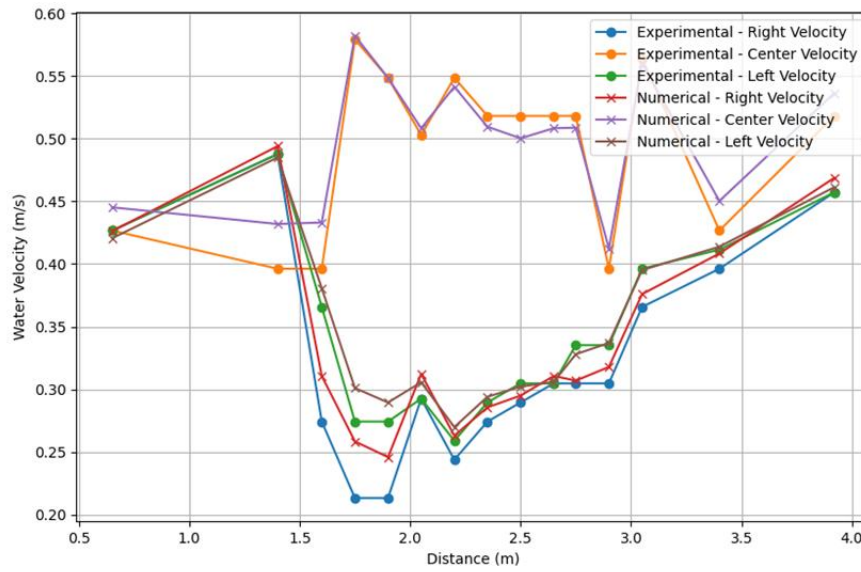


Figure 3.23. Variation of velocity for rectangular arc obstacles at $\dot{Q} = 0.0035 \text{ m}^3/\text{s}$, $S = 1.406^\circ$

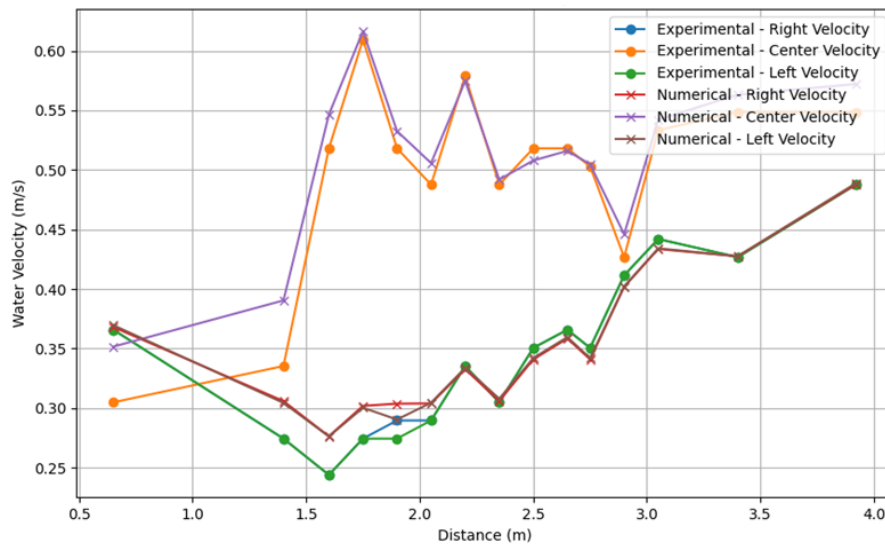


Figure 3.24. Variation of velocity for rectangular arc obstacles at $\dot{Q} = 0.004 \text{ m}^3/\text{s}$, $S = 1.406^\circ$

Likewise, Figure 3.25 shows the comparison between experimental and numerical water velocities around rectangular arch obstructions for $\dot{Q} = 0.0045 \text{ m}^3/\text{s}$ with a slope of 1.406° . At this higher slope, the increased gravitational force greatly enhances the interaction between the flow and the obstacles. This results in a noticeable increase in speed at the center of the channel within the obstacle area compared to the previous cases. Flow separation is more pronounced, with larger eddies forming downstream from the obstructions. Increased shear stress along the channel bed and walls exacerbates sediment transport and erosion processes. In addition, higher drag

forces contribute to greater energy dissipation. In general, increasing the flow rate with the highest slope of 1.406 degrees significantly affects the behavior of the velocity and its values in the presence of obstacles. Enhanced flow dynamics and obstacle interactions emphasize the critical role of slope and flow rate in shaping velocity, turbulence, and shear stress profiles in occluded channels. The average correlation coefficient, which reflects the strong agreement between experimental and numerical results, supports the ability of the numerical model to accurately simulate complex flow conditions.

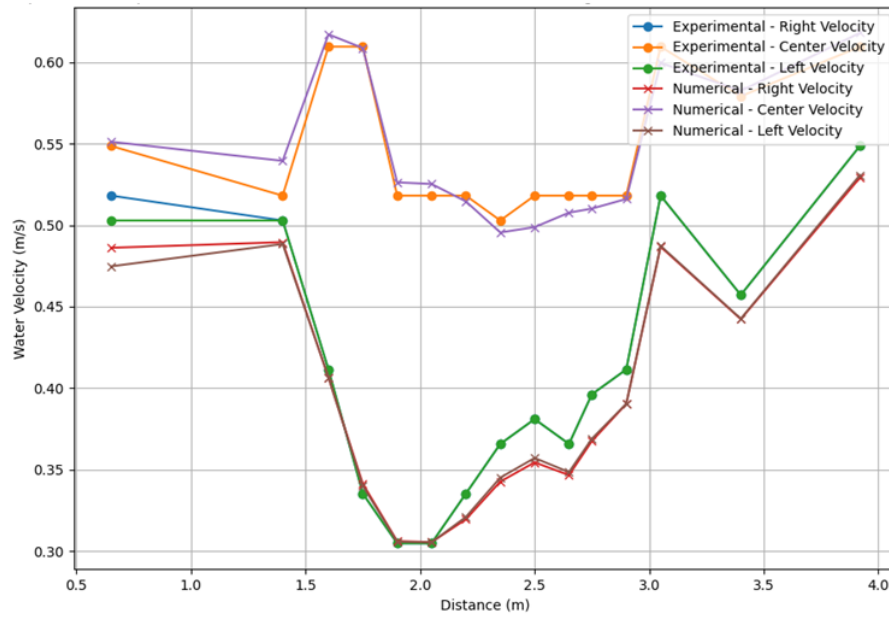


Figure 3.25. Variation of velocity for rectangular arc obstacles at $\dot{Q} = 0.0045 \text{ m}^3/\text{s}$, $S = 1.406^\circ$

Some numerical results of velocities and streamlines with rectangle arc obstacles will be presented now. When using obstructions in the form of rectangular plates with a curve at one end and a flow rate of $0.0035 \text{ m}^3/\text{s}$, as seen in Fig. 3.26 and Fig. 3.27, the velocity profiles before and after the obstructions are affected by slopes of 0° , 1.02° , and 1.406° as follows:

At a slope of 0° , the flow maintains a more uniform velocity along the channel. The presence of curved rectangular obstacles results in a significant decrease in velocity directly below the obstacle due to obstruction and flow separation. The curved end of the baffle reconnects the flow and makes for smoother transitions compared to baffles with sharp edges, slightly reducing turbulence and vortex formation.

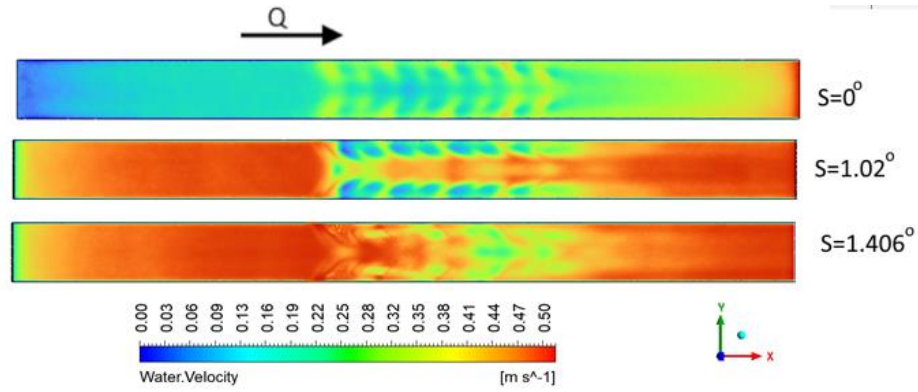


Figure 3.26. Numerical variation of velocity contours for rectangular arc obstacle $\dot{Q} = 0.0035 \text{ m}^3/\text{s}$

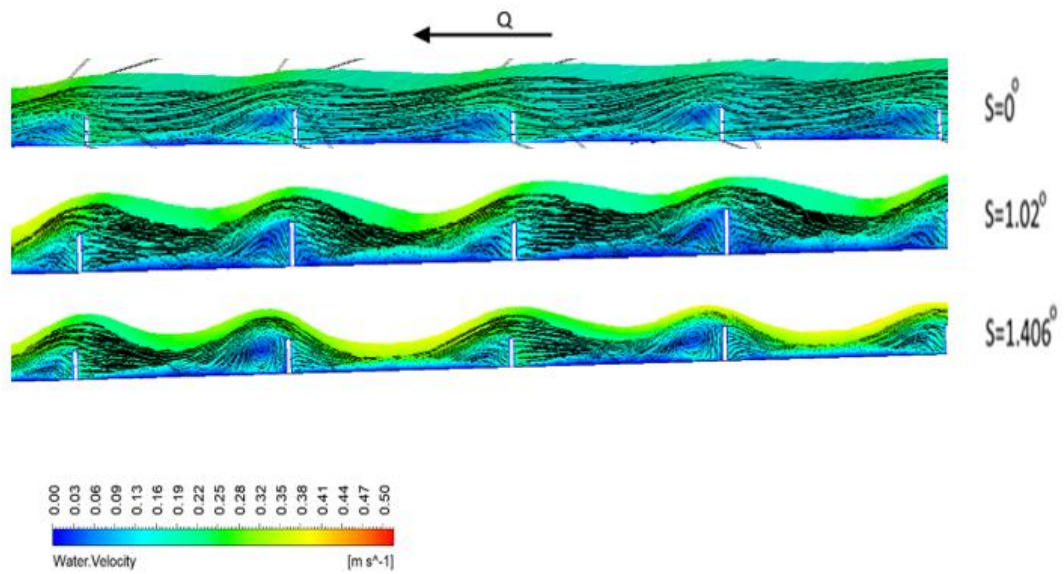


Figure 3.27. Numerical variation of stream lines for rectangular arc obstacle $\dot{Q} = 0.0035 \text{ m}^3/\text{s}$

With an incline of 1.02° , the initial speed before encountering obstacles increases due to the steeper incline. After passing curved rectangular obstacles, the velocity decreases significantly as the flow encounters increased resistance and separation zones. The curved ends of the baffles help smooth out some of the more abrupt changes in the flow direction, promoting a more gradual speed recovery and slightly reducing the intensity of the vortices compared to a 0° slope.

At the steepest slope of 1.406° , the initial velocity before obstacles is the highest due to the combined effects of steep slope and flow rate. Interaction with curved rectangular obstacles causes a sharp decrease in speed directly downstream. The curved ends of the baffles aid in smoother flow transitions, but the overall effect of a steep slope and high flow rate results in pronounced flow separation, high shear stress, and large vortex formation.

When using obstructions in the form of rectangular plates with a curve at one end and a flow rate of $0.004 \text{ m}^3/\text{s}$, Fig. 3.28 and Fig. 3.29 the velocity profiles before and after the obstructions are affected by slopes of 0° , 1.02° , and 1.406° as follows:

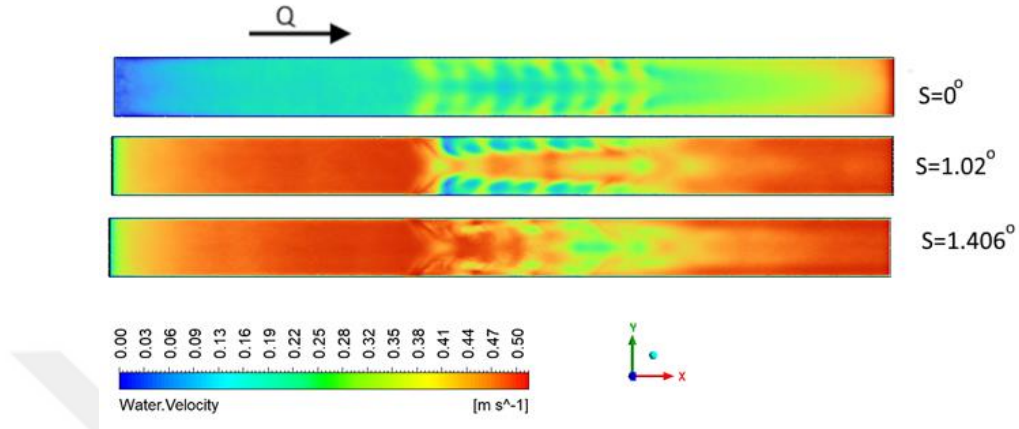


Figure 3.28. Numerical variation of velocity contours for rectangular arc obstacle $\dot{Q} = 0.004 \text{ m}^3/\text{s}$

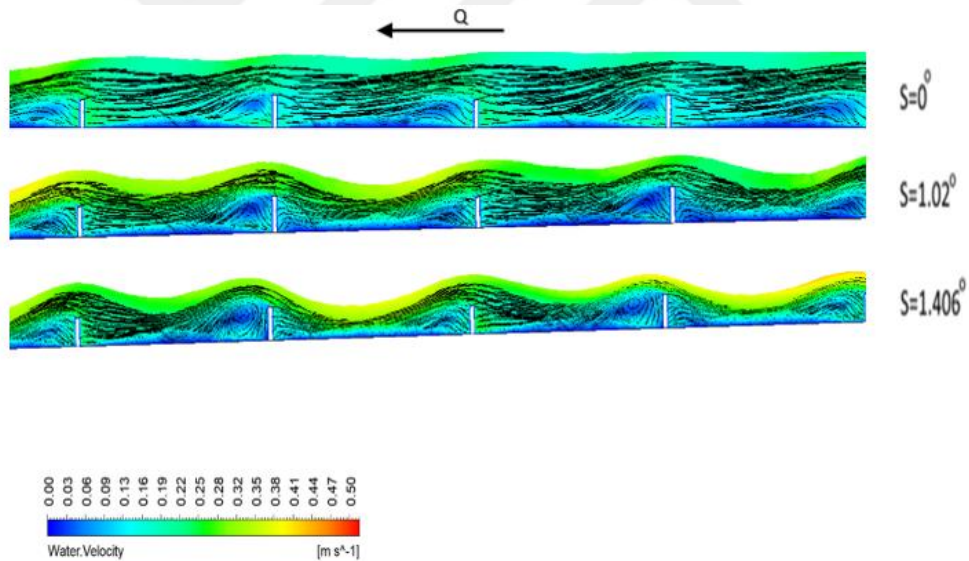


Figure 3.29. Numerical variation of stream lines for rectangular arc obstacle $\dot{Q} = 0.004 \text{ m}^3/\text{s}$

At a slope of 0° , the flow remains relatively stable and uniform along the channel before encountering obstructions. The velocity decreases significantly just below curved rectangular obstacles due to obstruction and flow separation. With a slope of 1.02° degrees, increasing the slope increases the initial velocity before the flow meets the obstacle. When interacting with curved rectangular obstacles, the speed decreases significantly due to increased resistance to flow and separation. At a slope of 1.406° , the initial velocity is higher due to the steeper slope and increased flow rate. The presence of curved rectangular obstacles causes a pronounced decrease

in velocity immediately downstream where the flow separates and forms eddies. When obstructions in the form of rectangular plates with a curve at one end are used and the flow rate is increased to $0.0045 \text{ m}^3/\text{s}$, Fig. 3.30 and Fig. 3.31 the velocity profiles before and after the obstructions for slopes of 0° , 1.02° , and 1.406° show the following behavior.

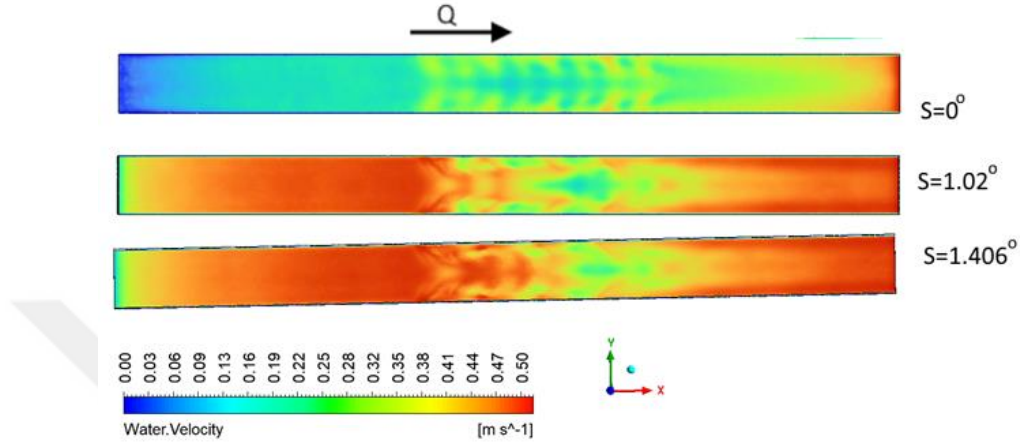


Figure 3.30. Numerical variation of velocity contours for rectangular arc obstacle $\dot{Q} = 0.0045 \text{ m}^3/\text{s}$

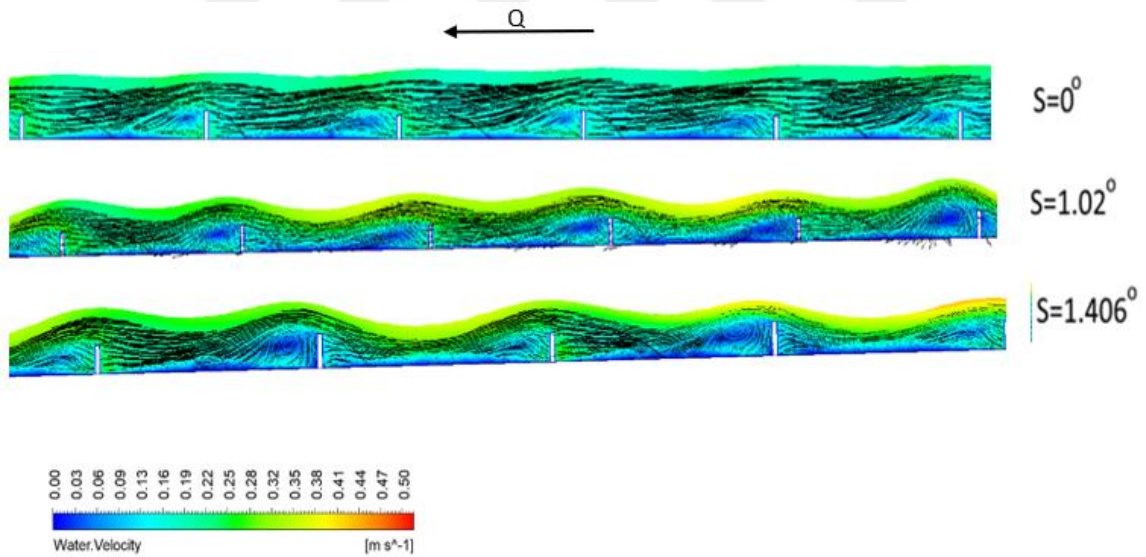


Figure 3.31. Numerical variation of stream lines for rectangular arc obstacle $\dot{Q} = 0.0045 \text{ m}^3/\text{s}$

At a slope of 0° , the flow remains relatively stable before encountering obstacles due to the absence of gravity. When encountering curved rectangular obstacles, the speed decreases significantly due to flow obstruction and separation. The curved ends of the baffles help mitigate sudden changes in flow direction, allowing flow to be reconnected more smoothly downstream. However, a high flow rate exacerbates the velocity drop and slightly increases turbulence compared to low flow rates.

With a slope of 1.02° , the slight tilt increases the initial speed of the water before it hits obstacles. Interaction with curved rectangular obstacles results in a significant reduction in velocity directly downstream due to increased flow resistance and separation. The curved tips facilitate smoother transitions and aid in the gradual recovery of velocity downstream, although a higher flow rate results in more pronounced vortex formation and turbulence compared to lower flow rates.

At a slope of 1.406° , a steeper slope results in a higher initial water velocity before it encounters obstacles. The presence of curved rectangular obstacles causes a significant decrease in velocity immediately downstream where the flow separates and forms eddies. Curved tips help mitigate some turbulence by allowing smoother flow transitions, but the combined effect of high slope and increased flow rate results in significant flow turbulence. This leads to pronounced shear stresses and complex flow patterns around obstacles.

In summary, increasing the flow rate to $0.0045 \text{ m}^3/\text{s}$ for curved rectangular plate obstacles results in more pronounced changes in the velocity profiles at all slopes. At 0° degrees, the flow remains relatively stable with smoother transitions due to the curved ends of the obstructions, but the higher flow rate increases flow separation and turbulence. At 1.02° , the higher velocity before the obstacles results in significant flow separation and turbulence, which is partially mitigated by the curved tips. At 1.406° , a steeper slope with a higher flow rate results in higher initial velocities and more complex flow turbulence, with significant shear stress and pronounced vortex formation despite the curved ends aiding smoother transitions. The behavior of velocity profiles is greatly influenced by the combination of increased flow rate and slope, resulting in more complex and turbulent flow patterns around obstacles.

3.1.3. Velocity Variations along Channel Distance with Cube Obstacles

When cube obstacles are used under the conditions of the $\dot{Q} = 0.0035 \text{ m}^3/\text{s}$, $S = 0^\circ$, the velocity variations are plotted as in Fig. 3.32. Both sets of data exhibit similar trends in velocity behavior along the distance.

Overall, there is reasonable agreement between experimental and numerical results, indicating a somewhat satisfactory simulation of the fluid flow around cube obstacles. The presence of cube obstacles causes variations in velocity profiles, albeit less pronounced compared to more complex shapes. Nevertheless, these variations can still lead to the formation of separation zones and eddies. Additionally, the distribution of shear stress on the surface of obstacles and surrounding areas is affected by these velocity variations. Drag force experienced by the obstacles may also be influenced, though to a lesser extent compared to more streamlined shapes.

Figure 3.33 shows the velocity distributions when increasing the flowrate to $\dot{Q} = 0.004$ m³/s, but still with the same slope, $S = 0^\circ$. The general trend seen in Fig. 3.32 is seen in Fig. 3.33. The distribution of shear stress on the surface of obstacles and surrounding areas is affected by the velocity variations

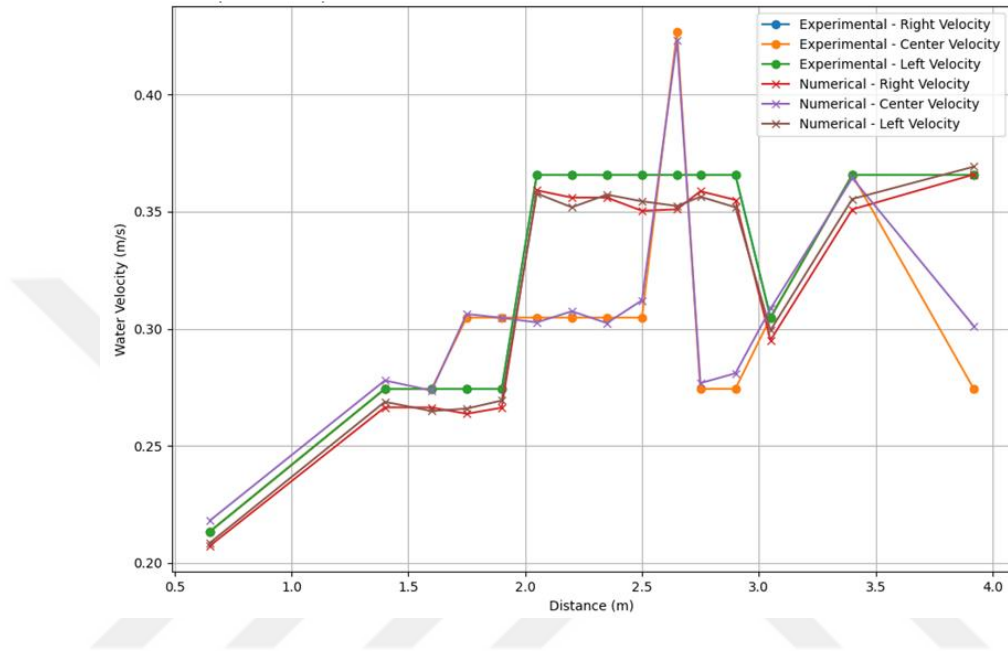


Figure 3.32. Variation of velocity for cube obstacles at $\dot{Q} = 0.0035$ m³/s, $S = 0^\circ$

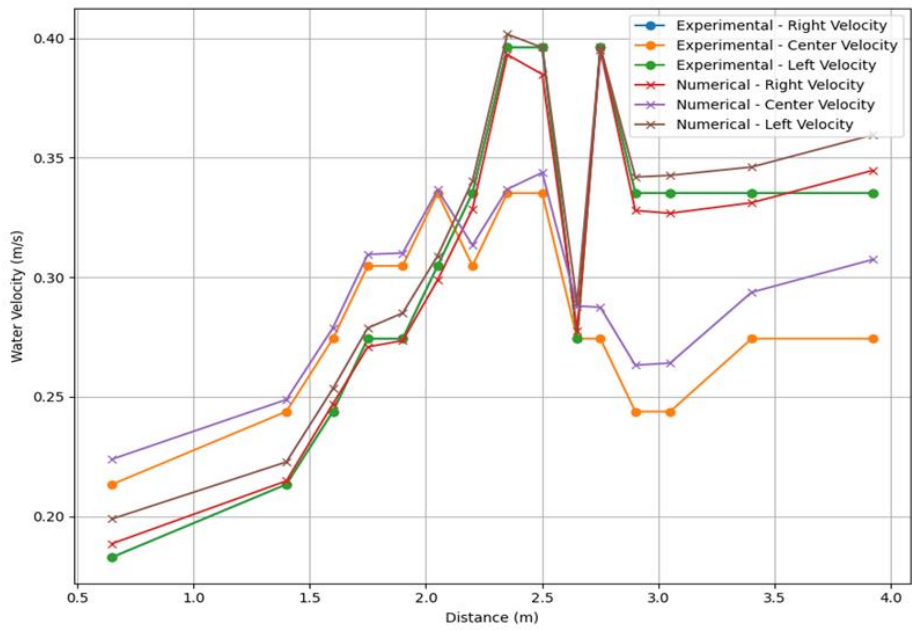


Figure 3.33. Variation of velocity for cube obstacles at $\dot{Q} = 0.004$ m³/s, $S = 0^\circ$

Figure 3.34 shows the case of $\dot{Q} = 0.0045 \text{ m}^3/\text{s}$, $S = 0^\circ$. There is also a reasonable agreement between experimental and numerical results. All comments that are valid for the Fig. 3.32 and Fig. 3.33 are still valid for this case.

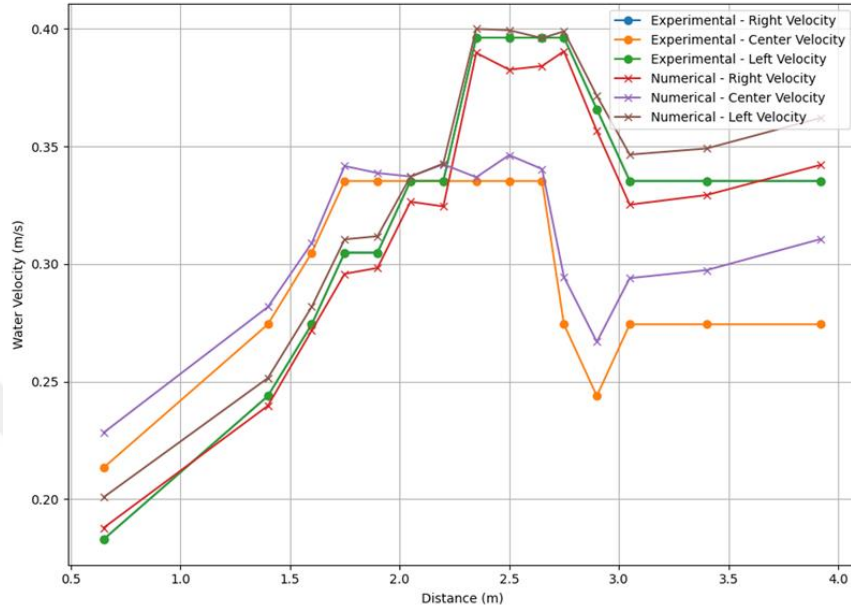


Figure 3.34. Variation of velocity for cube obstacles at $\dot{Q} = 0.0045 \text{ m}^3/\text{s}$, $S = 0^\circ$

Figure 3.35 presents the case of $\dot{Q} = 0.0035 \text{ m}^3/\text{s}$, $S = 1.0156^\circ$. Both experimental and numerical results exhibit similar trends in velocity behavior along the distance.

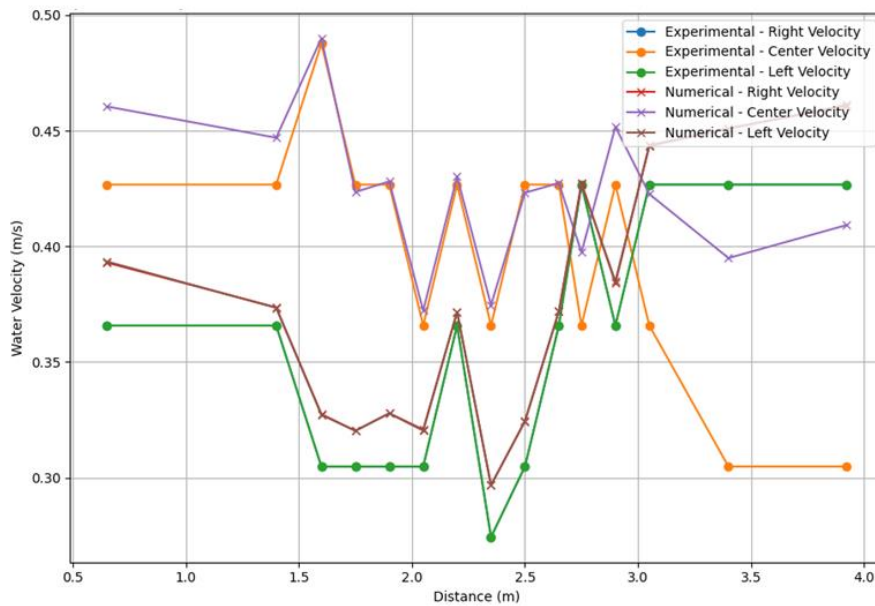


Figure 3.35. Variation of velocity for cube obstacles at $\dot{Q} = 0.0035 \text{ m}^3/\text{s}$, $S = 1.0156^\circ$

The plotted data in Fig. 3.36 illustrates the comparison between experimental and numerical water velocities around cube obstacles under the conditions of $\dot{Q} = 0.004 \text{ m}^3/\text{s}$, $S = 1.0156^\circ$. The plot in Fig. 3.37 illustrates the variations for $\dot{Q} = 0.0045 \text{ m}^3/\text{s}$, $S = 1.0156^\circ$.

All physical explanations for Fig. 3.35 and Fig. 3.36 are valid for Fig. 3.37. However, to keep away from repetitive explanations some comments are not repeated here. Although the velocity profiles exhibit similar trends, discrepancies in magnitude are noticeable, particularly at certain distances. The presence of cube obstacles disrupts the flow, leading to deviations in the velocity distribution and contributing to flow separation. Eddies or vortices form behind the obstacles, enhancing mixing and redistributing momentum within the flow.

Moreover, the obstacles increase shear stress on the channel bed and walls and experience drag forces as water flows around them. These interactions between the obstacles and flowing water influence flow dynamics, impacting separation, eddies formation, shear stress, and drag force. The average correlation coefficient between experimental and numerical results quantifies the agreement, providing insight into the accuracy of the numerical model in predicting flow behavior. Overall, the comparison sheds light on the complex flow phenomena influenced by cube obstacles and their implications for various hydraulic parameters.

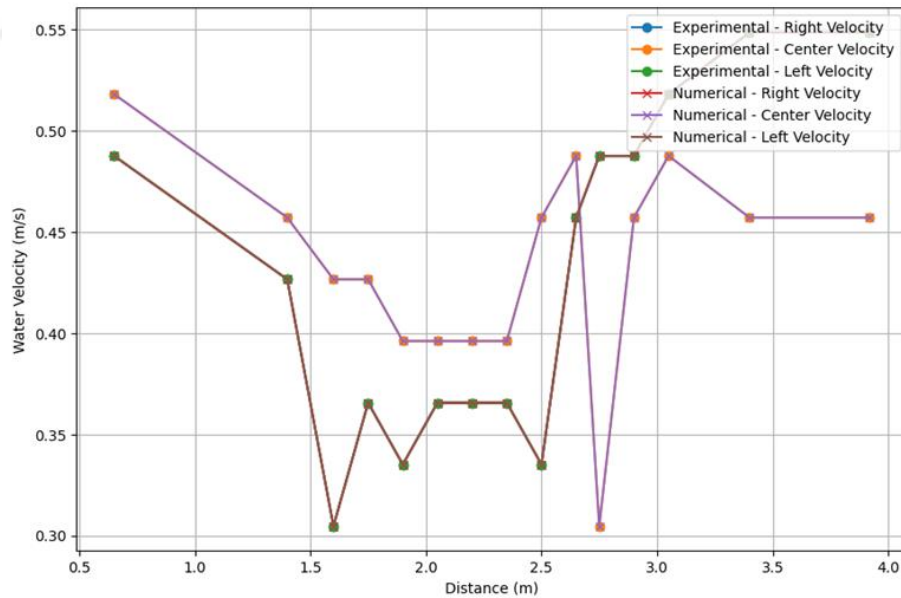


Figure 3.36. Variation of velocity for cube obstacles at $\dot{Q} = 0.004 \text{ m}^3/\text{s}$, $S = 1.0156^\circ$

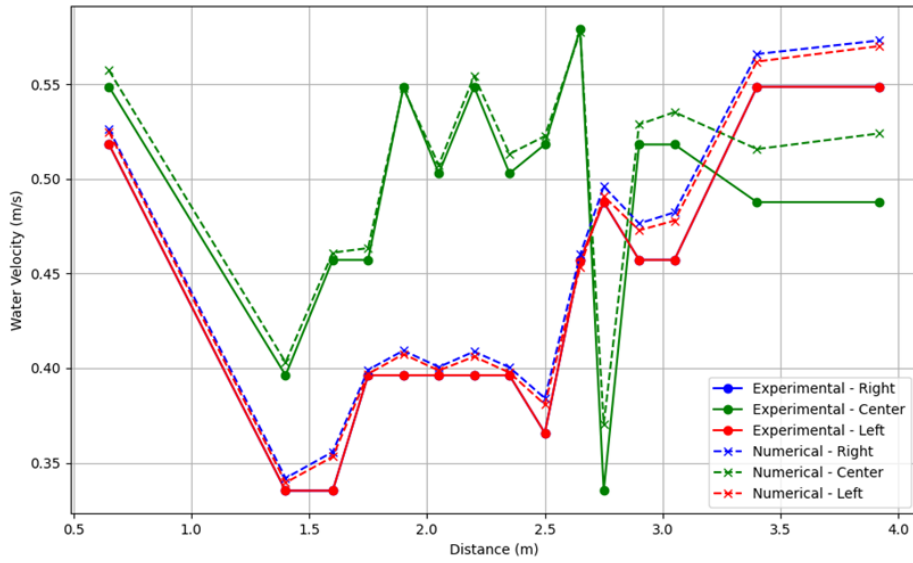


Figure 3.37. Variation of velocity for cube obstacles at $\dot{Q} = 0.0045 \text{ m}^3/\text{s}$, $S = 1.0156^\circ$

Figure 3.38 presents the case of $\dot{Q} = 0.0035 \text{ m}^3/\text{s}$, $S = 1.406^\circ$. The variations in velocity indicate the influence of obstacles on flow behavior. Differences between the two datasets highlight discrepancies in the numerical model's ability to accurately predict flow characteristics. Separation zones and eddies formation behind the obstacles are observed, affecting velocity profiles. Additionally, variations in shear stress and drag force can be inferred from the velocity distribution.

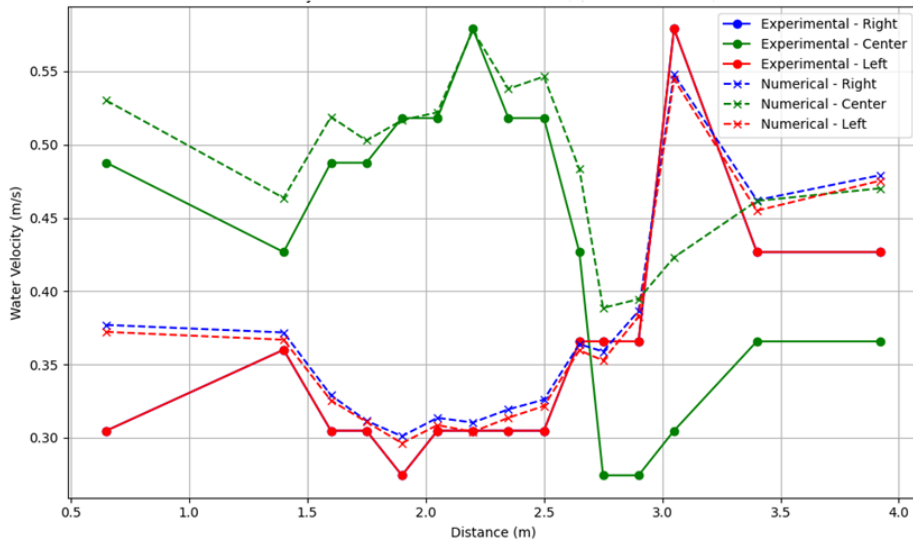


Figure 3.38. Variation of velocity for cube obstacles at $\dot{Q} = 0.0035 \text{ m}^3/\text{s}$, $S = 1.406^\circ$

As shown in Fig 3.39, for the case $\dot{Q} = 0.004 \text{ m}^3/\text{s}$, $S = 1.406^\circ$, comparing the experimental and numerical results of water velocity for cube obstacles reveals several interesting observations. In general, both the experimental and numerical data show similar trends in water velocity distribution along the length of the flow field. At the right side of the obstacle (the region where the flow encounters the obstacle), both experimental and numerical data indicate a decrease in water velocity compared to the center and left regions. This reduction in velocity is expected as the flow interacts with the obstacle, causing obstruction and hence reducing the flow velocity. In the center region, where the flow passes through the gap between obstacles, both experimental and numerical data show relatively higher water velocities compared to the right and left regions. This is consistent with the flow behavior around obstacles, where the velocity tends to increase in regions of reduced obstruction. On the left side of the obstacle (downstream of the flow), both experimental and numerical results show a slight decrease in water velocity compared to the center region. This decrease can be attributed to the flow recovery after passing through the obstacle, where the velocity gradually returns to its original state. Overall, the comparison between experimental and numerical results demonstrates a good agreement, indicating that the numerical simulation accurately captures the flow behavior around cube obstacles. The average correlation coefficient between the experimental and numerical data can provide a quantitative measure of this agreement, indicating the reliability of the numerical model in predicting the flow characteristics.

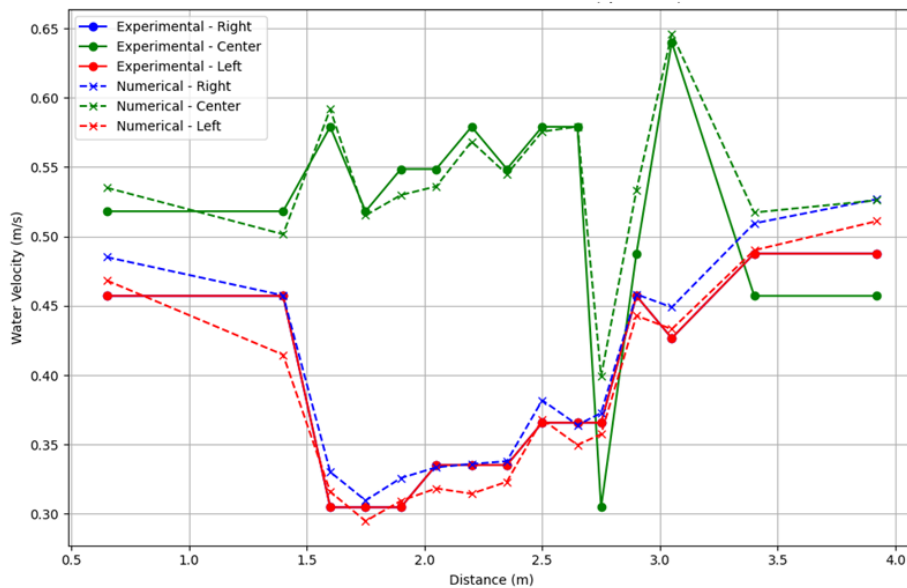


Figure 3.39. Variation of velocity for cube obstacles at $\dot{Q} = 0.004 \text{ m}^3/\text{s}$, $S = 1.406^\circ$

The observed changes in water velocity due to the presence of cube obstacles have implications for various flow phenomena, including separation, eddies formation, shear stress, and drag force. Understanding these effects is crucial for applications such as fluid dynamics, hydrodynamics, and hydraulic engineering, where accurate prediction of flow behavior is essential for design and analysis purposes.

Figure 3.40 is plotted to compare experimental and numerical results of water velocity for cube obstacles with a flow rate $\dot{Q} = 0.0045 \text{ m}^3/\text{s}$ and a slope $S = 1.406^\circ$ reveals interesting insights into the behavior of the flow.

Both the experimental and numerical data exhibit similar trends along the length of the flow field. In the region immediately to the right of the obstacles, both experimental and numerical results show a decrease in water velocity compared to the center and left regions. This reduction in velocity is a consequence of the flow encountering the obstacle, resulting in obstruction and hence a decrease in flow velocity.

In the middle region, where the flow passes through the gap between obstacles, experimental and numerical data indicate relatively higher water velocities compared to the right and left regions. This behavior is expected, as obstructions caused by obstructions in the center are reduced, allowing for higher flow velocities. On the left side of the obstruction (bottom of the flow), experimental and numerical results show a slight decrease in water velocity compared to the central region. This decrease can be attributed to the recovery of flow after passing through obstacles, as the speed gradually returns to its original state.

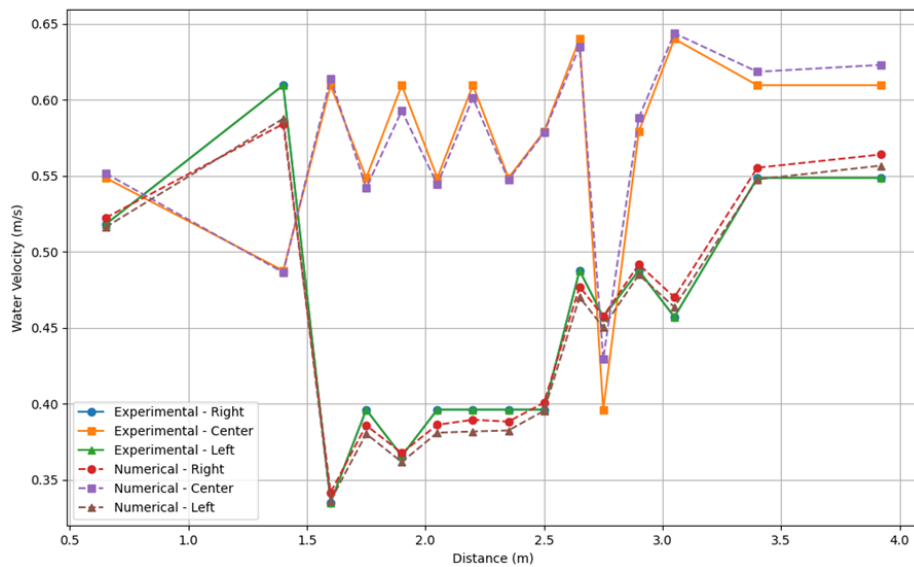


Figure 3.40. Variation of velocity for cube obstacles at $\dot{Q} = 0.0045 \text{ m}^3/\text{s}$, $S = 1.406^\circ$

Figures 3.41 and 3.42 show the numerical behaviors of the velocity profiles before and after cubic obstacles arranged on either side of the channel, affected by a flow rate of $0.0035 \text{ m}^3/\text{s}$ with slopes of 0° , 1.02° , and 1.406° . At a slope of 0° , the velocity profile shows a more uniform distribution with relatively lower velocities before obstacles and a modest decrease after obstacles due to minimal gravitational influence. At a slope of 1.02° , the velocity before the obstacle increases due to the steeper slope, resulting in a more pronounced decrease in velocity after the obstacle as the flow encounters a greater obstruction. At a steeper slope of 1.406° , the initial velocities are higher, resulting in a significant reduction in velocity past the obstacle due to enhanced gravitational acceleration and increased flow resistance. These differences in slope affect velocity gradients, flow separation, vortex formation, shear stress, and drag forces around obstacles, with higher slopes intensifying these effects.

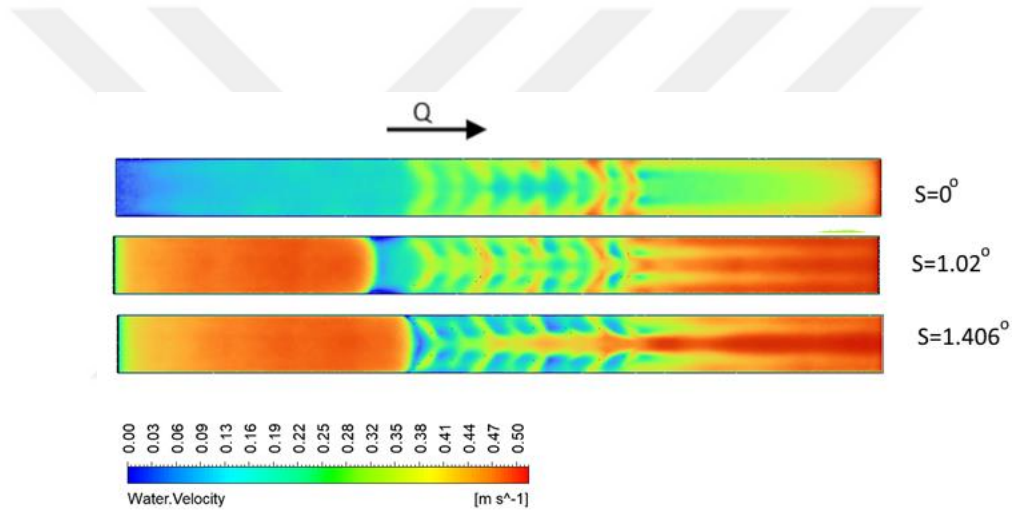


Figure 3.41. Numerical variation of velocity contours for cubic obstacle $\dot{Q} = 0.0035 \text{ m}^3/\text{s}$

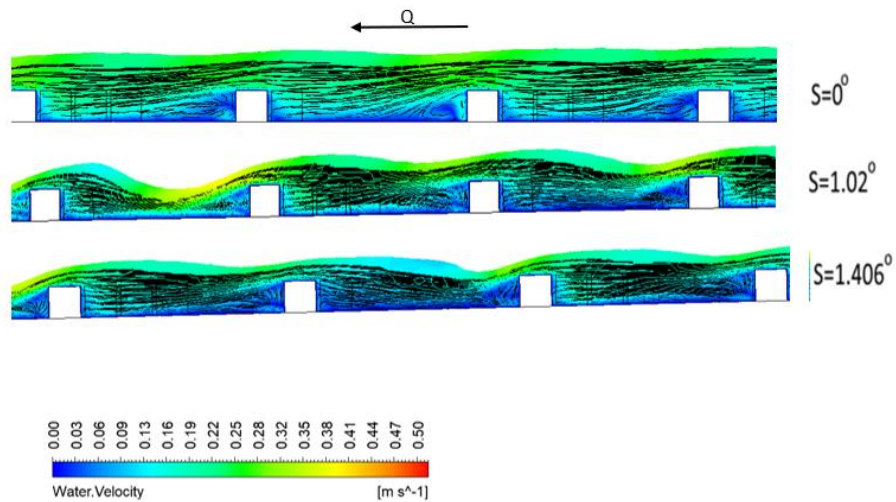


Figure 3.42. Numerical variation of stream lines for cubic obstacle $\dot{Q} = 0.0035 \text{ m}^3/\text{s}$

When the flow rate is increased to $0.004 \text{ m}^3/\text{s}$ for the same cubic obstructions and slopes of 0° , 1.02° , and 1.406° , Fig. 3.43 and Fig. 3.44 are obtained. At a slope of 0° , the uniform distribution of velocity becomes slightly more pronounced, with a moderate increase before obstacles and a marked decrease after obstacles due to increased flow resistance.

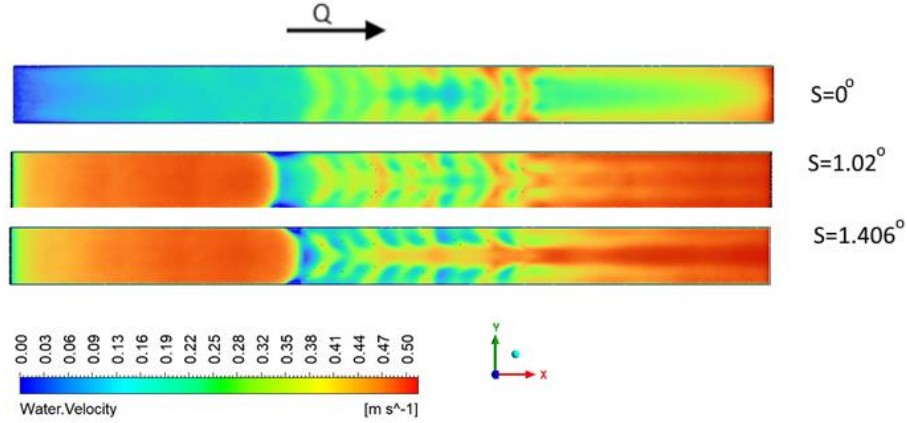


Figure 3.43. Numerical variation of velocity contours for cubic obstacle $\dot{Q} = 0.004 \text{ m}^3/\text{s}$

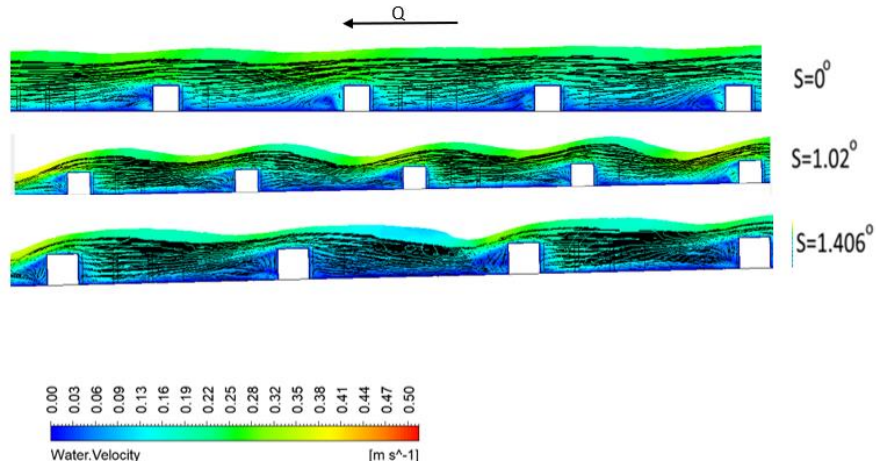


Figure 3.44. Numerical variation of stream lines for cubic obstacle $\dot{Q} = 0.004 \text{ m}^3/\text{s}$

At a slope of 1.02° , the velocity increases significantly before the obstacle due to the steeper slope and higher flow rate, resulting in a greater decrease in velocity after the obstacle as the flow encounters an increasing obstacle. At the steep slope of 1.406° , the initial velocities are much higher due to the combined effect of the steep slope and increased flow rate, which results in a significant decrease in velocity past the obstacle due to enhanced gravitational acceleration and increased flow resistance. When the flow rate is increased to $0.0045 \text{ m}^3/\text{s}$ for the same obstacles and cubic slopes of 0° , 1.02° , and 1.406° , Figures 3.45 and 3.46 show the velocity profiles.

At a slope of 0° , the velocity increases uniformly along the channel, but the presence of obstacles still causes a significant decrease in velocity directly downstream due to increased flow resistance. These differences in slope and flow rate significantly affect velocity behavior and flow dynamics, especially around obstacles, by changing velocity gradients, flow separation, vortex formation, shear stress, and drag forces. Higher slopes and flow rates intensify these effects, leading to more pronounced changes in flow characteristics.

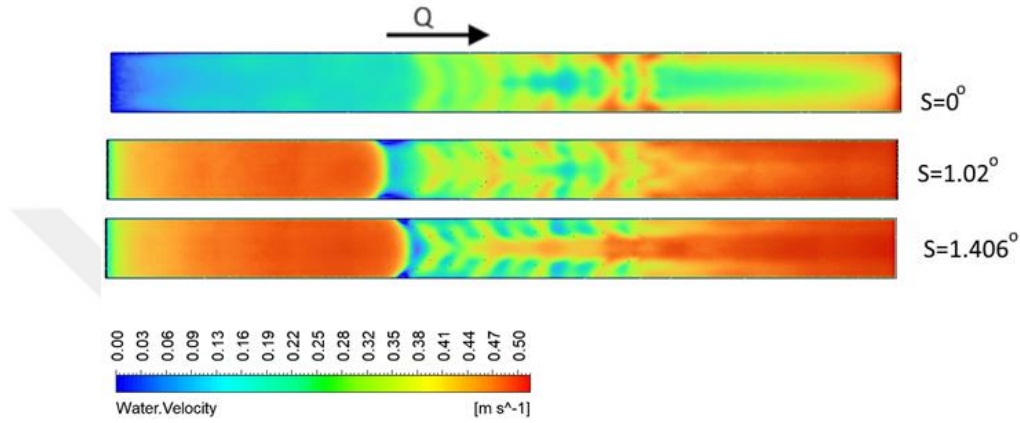


Figure 3.45. Numerical variation of velocity contours for cubic obstacle $\dot{Q} = 0.0045 \text{ m}^3/\text{s}$

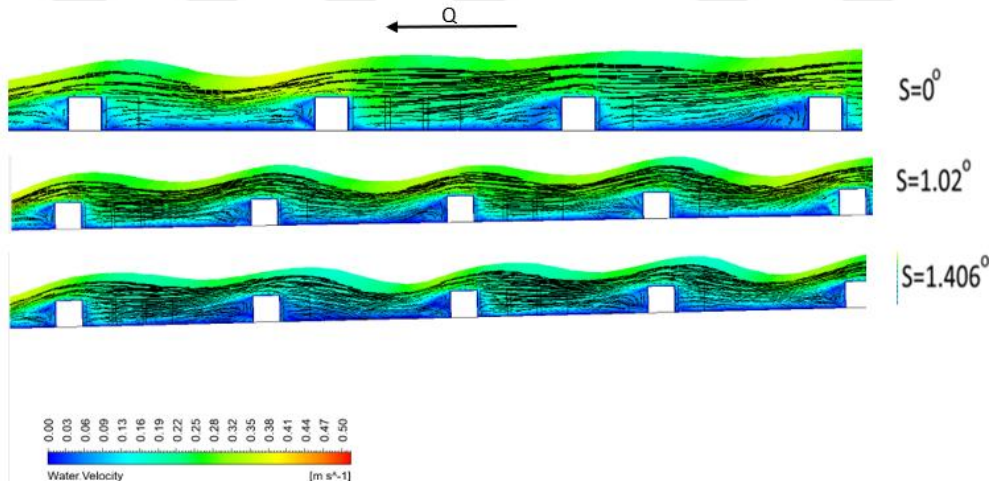


Figure 3.46. Numerical variation of stream lines for cubic obstacle $\dot{Q} = 0.0045 \text{ m}^3/\text{s}$

With an inclination of 1.02° , the speed before obstacles increases more sharply due to the steeper incline and higher flow rate. After passing obstacles, the speed drops dramatically, creating clear separation zones and vortices. At the steeper slope of 1.406° , the velocity profiles exhibit the highest initial velocities due to the combined effects of the steep slope and high flow rate. This results in a significant decrease in velocity past obstacles, with significant flow separation, increased shear stress, and pronounced vortex formation

3.1.4. Velocity Variations along Channel Distance with Circular Obstacles (Experimental Only)

Velocity profiles were analyzed in an open channel provided by cylindrical obstacles arranged on either side of the channel for slopes of 0° , 1.02° , and 1.406° . The velocity variations are shown in Fig. 3.47, Fig. 3.48, and Fig. 3.49 respectively for the flowrates of $0.0035 \text{ m}^3/\text{s}$, $0.004 \text{ m}^3/\text{s}$, and $0.0045 \text{ m}^3/\text{s}$.

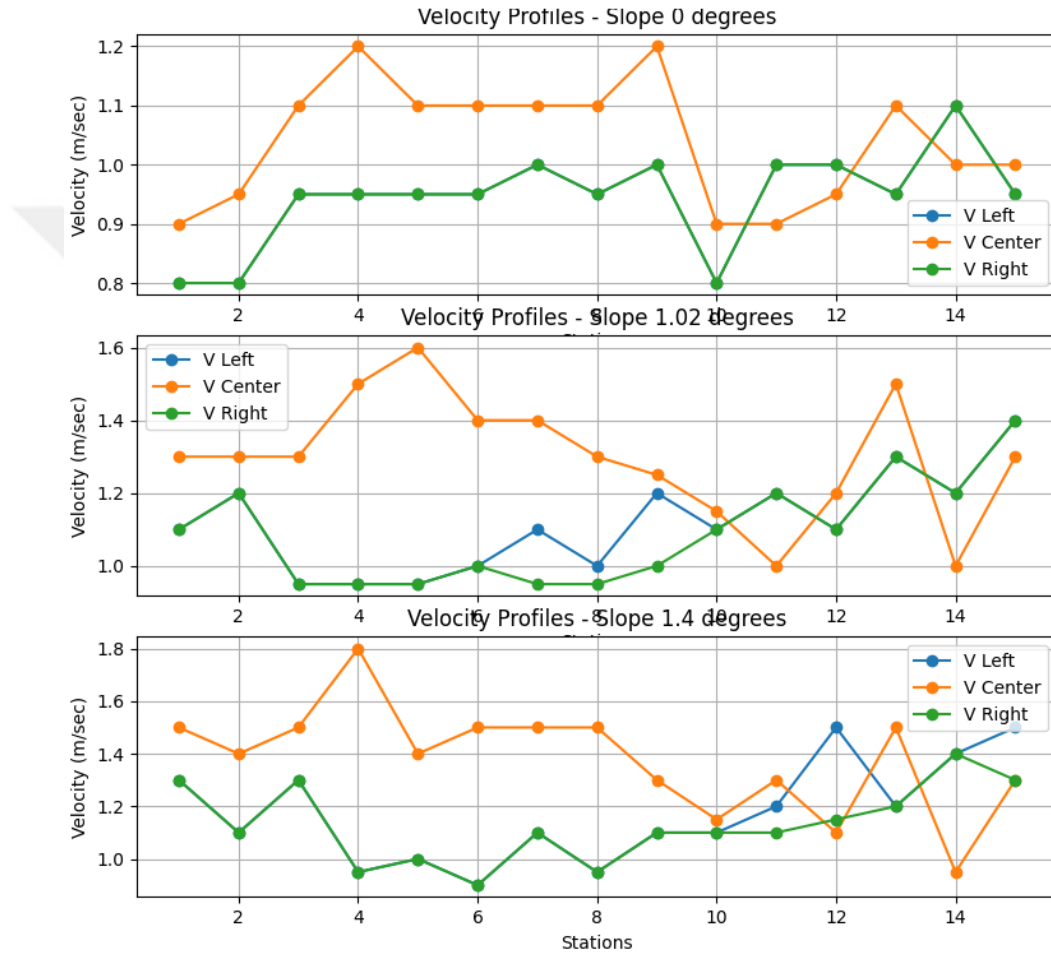


Figure 3.47. Variation of velocity for cylindrical obstacle $\dot{Q} = 0.0035 \text{ m}^3/\text{s}$

Regardless of the flow rate, for a 0° slope, the velocity profiles show differences between the left (VL), middle (VC), and right (VR) sections at different stations along the channel. In general, velocities are relatively consistent across sections, with minor fluctuations observed. According to all three figures, at a slope of 1.02° , the velocity profiles show more pronounced differences between the left, middle, and right sections than at a slope of 0° . Some stations show significant increases or decreases in velocity, especially in the central section, indicating increased flow disturbances caused by cylindrical obstructions.

For the steeper slope of 1.406° , the velocity profiles display more significant differences between the left, middle and right sections, with greater discrepancies in observed velocities across different stations. This indicates more flow disturbances and significant changes in velocity patterns due to increased slope and the presence of cylindrical obstructions. Plotting the velocity curves for each set of data (0° slope, 1.0156° slope, and 1.406° slope) allows a visual representation of the velocity behavior at different sections along the channel. The curves show how the velocity profiles change with different slopes and the effect of cylindrical obstacles on the flow dynamics.

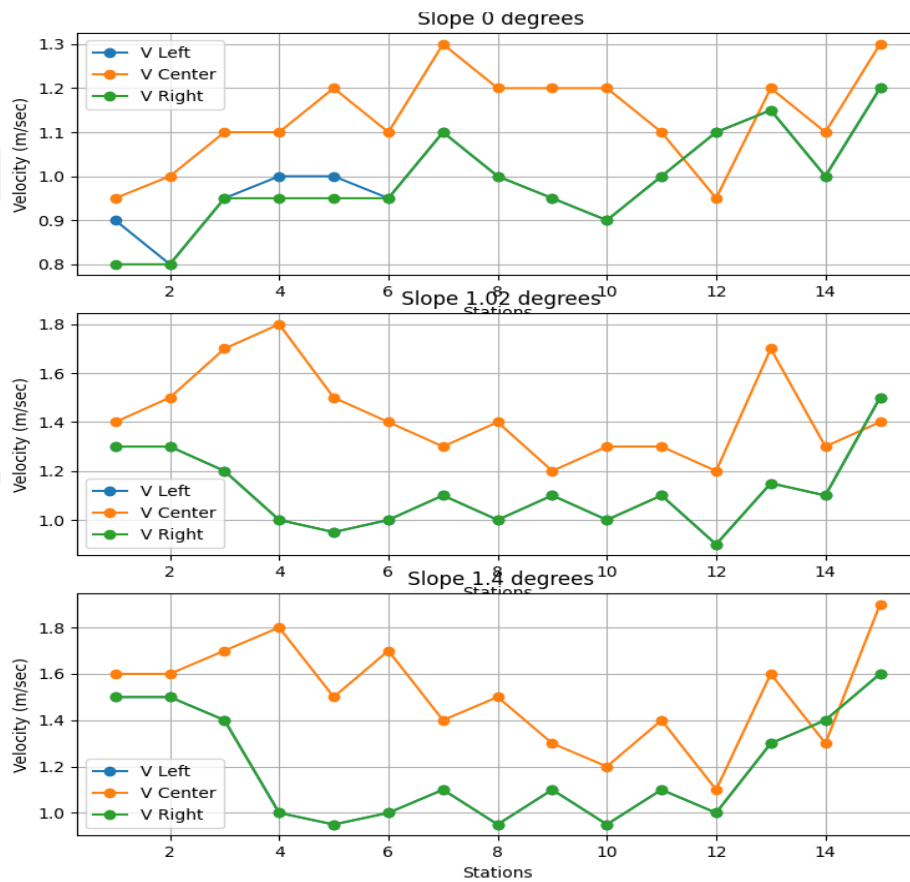


Figure 3.48. Variation of velocity for cylindrical obstacle $\dot{Q} = 0.004 \text{ m}^3/\text{s}$

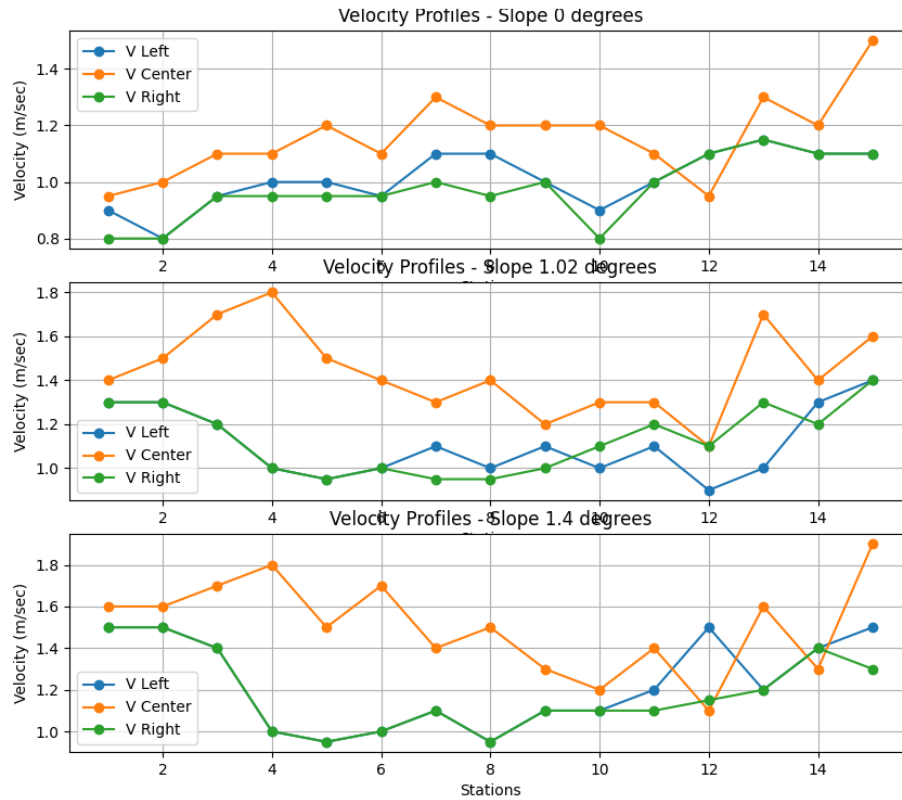


Figure 3.49. Variation of velocity for cylindrical obstacle $\dot{Q} = 0.0045 \text{ m}^3/\text{s}$

3.1.5. Velocity Variations along Channel Distance with Elliptic Obstacles (Experimental Only)

The flow velocity data for the three different slopes (0° , 1.02° , and 1.406°) with a constant discharge of $\dot{Q} = 0.0035 \text{ m}^3/\text{s}$ show consistent trends highlighting the effect of slope on flow velocity in open channels (Fig. 3.50). At a 0° slope, flow velocities are relatively lower, indicating slower water flow, with fairly constant velocities for the left, middle, and right sections across most stations. As the slope increases to 1.02° , flow velocities show a significant increase compared to the 0° slope, indicating the effect of a steeper slope facilitating faster water movement. The 1.406° slope, the steepest of the three, results in the highest flow velocities, reflecting the most significant effect on flow. This consistent pattern across different slopes confirms the basic principle that increasing slope leads to higher flow velocities due to the force of gravity acting more strongly along the flow direction. This understanding is critical to the design and management of water conveyance systems, such as irrigation canals, drainage canals, and river engineering projects, to ensure efficient flow and prevent problems such as erosion or sediment deposition. By considering the relationship between slope and flow velocity, engineers can improve the design of these systems, reduce maintenance requirements, and increase their effectiveness in transporting water and sediment.

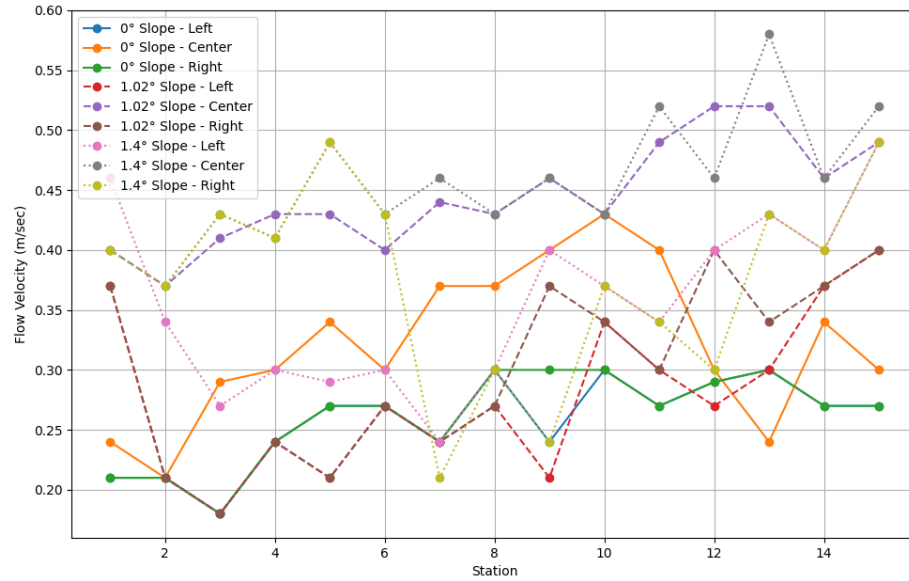


Figure 3.50. Variation of water flow velocity for elliptical obstacle $Q = 0.0035 \text{ m}^3/\text{s}$

The plotted graphs offer valuable insights into the behavior of flow velocities at different stations for various slopes with a discharge of $0.004 \text{ m}^3/\text{s}$ for Newtonian flow around an elliptical obstacle (Fig. 3.51). The observations reveal significant variations in flow velocities across all slopes (0° , 1.02° , and 1.406°), with velocities generally increasing initially, peaking at certain stations, and then stabilizing or slightly decreasing towards the end. When comparing the slopes, the 0° slope (Solid Line) shows relatively consistent flow velocities for the left, center, and right sections, ranging from approximately 0.24 to 0.46 m/s. In contrast, the 1.02° slope (Dashed Line) exhibits higher velocities, especially in the center section where peaks are more pronounced, ranging from around 0.21 to 0.58 m/sec. The 1.406° slope (Dotted Line) displays the highest velocities, with substantial increases in the right section and peaks at stations 9 and 13, ranging from approximately 0.24 to 0.58 m/sec. Section-wise observations highlight the trends in each section, with the left section showing a steady increase in velocity with slope, the center section consistently having the highest velocities, and the right section mirroring the left but with slightly higher velocities in some cases. These observations have implications for flow management, emphasizing the importance of understanding flow velocities at different slopes for designing hydraulic structures and managing water flow efficiently. The data suggests that steeper slopes lead to higher velocities, impacting erosion rates, sediment transport, and channel stability, crucial considerations for flood control and water management systems. In conclusion, the study underscores how slope gradient significantly influences flow characteristics, with steeper slopes resulting in higher and more variable velocities. This understanding is essential for maintaining stable and efficient water flow in natural and engineered channels, highlighting the intricate

relationship between slope, flow velocity, and station position for practical applications in water resource management.

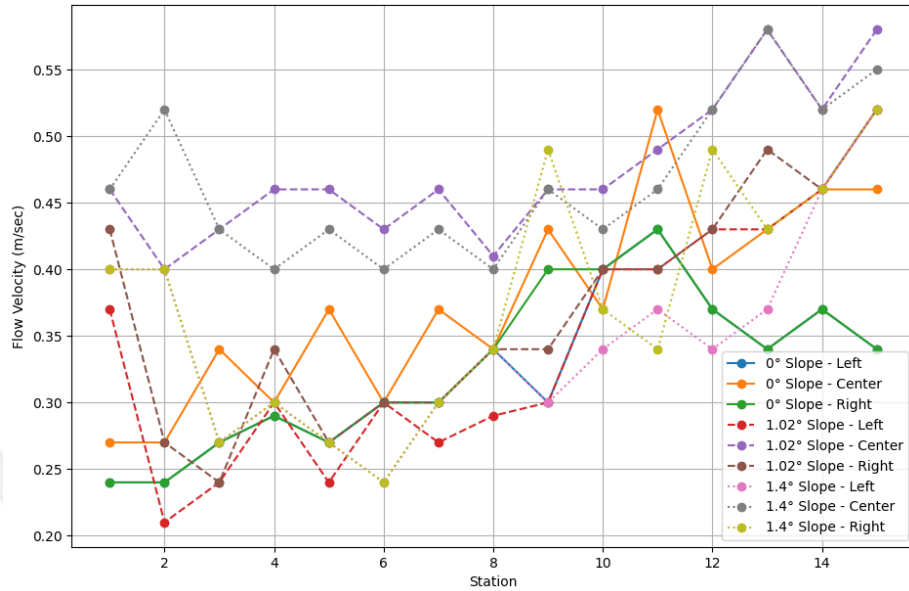


Figure 3.51. Variation of water flow velocity, Elliptical obstacle, $\dot{Q} = 0.004 \text{ m}^3/\text{s}$

In Fig. 3.52, flow velocities vary significantly at different stations across all slopes (0° , 1.02° , and 1.406° , $\dot{Q} = 0.0045 \text{ m}^3/\text{s}$), with an initial increase, peaking at certain stations, and then stabilizing or decreasing slightly toward the end. When comparing slopes, a 0° slope shows the lowest velocities, indicating a more consistent flow pattern with less resistance. In contrast, a slope of 1.02° shows a significant increase in velocities, especially in the middle section where the peaks are more pronounced. The 1.406° slope shows the highest velocities of the three, with a significant increase in velocities, especially in the right section, reflecting more turbulent and accelerated flows due to the steeper gradient. Observations in each section reveal a steady increase in velocity with a slope in the left section, the middle section consistently experiences the highest velocities, and the right section shows slightly higher velocities in some cases, with a significant increase in flow velocity at the 1.406° slope. Understanding flow velocities at different slopes is crucial to designing hydraulic structures and managing water flow efficiently. Steeper slopes resulting in higher velocities can affect erosion rates, sediment transport, and channel stability, necessitating consideration of flood control and water management planning. In conclusion, the study highlights how slope gradient significantly affects flow characteristics, with steeper slopes resulting in higher and more variable velocities. This understanding is essential to maintaining stable and efficient water flow in natural and engineered channels, emphasizing the importance of slope management in hydraulic and environmental engineering. The complex

relationship between slope, flow velocity, and station position provides valuable insights for practical applications in water resources management.

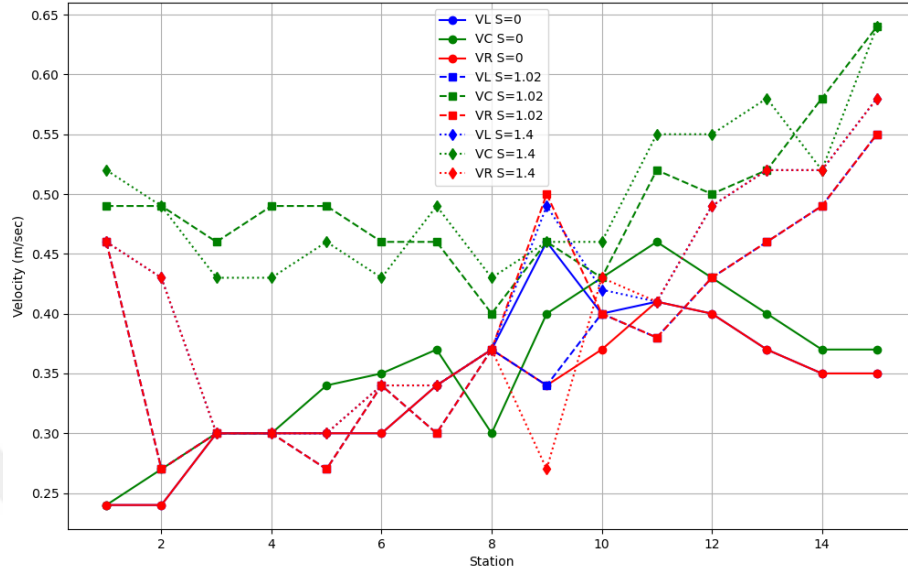


Figure 3.52. Variation of water flow velocity ,Elliptical obstacle, $\dot{Q} = 0.0045 \text{ m}^3/\text{s}$

3.2. Comparison of the Slope on the Velocity

Overall, the results highlight the complex relationship between obstacle geometry and fluid velocity, emphasizing the need for comprehensive consideration of slope effects in fluid dynamics analyses. Such comprehensive analyses are essential for understanding and predicting fluid flow phenomena across different flow rates and geometric configurations, contributing to advances in practical applications.

The streamlines in Fig. 3.53a and 3.53b show the effect of slope with obstacles. Fig. 3.54 presents the water profiles from the top view with slope $S = 1.406^\circ$, $\dot{Q} = 0.0045 \text{ m}^3/\text{s}$. Fig 3.55 presents the depth profile from a side view with a slope for $S = 0^\circ$, 1.02° , 1.406° respectively ($\dot{Q} = 0.0035 \text{ m}^3/\text{s}$). Depth profile in the first obstacle at different slopes, explaining the flood jump for $S = 1.02^\circ$ and $\dot{Q} = 0.0035 \text{ m}^3/\text{s}$ is given in Fig. 3.56. Waterside profile at the first obstacle, $S = 1.406^\circ$ and $\dot{Q} = 0.004 \text{ m}^3/\text{s}$ compatibility between experiment and numerical results is given in Fig. 3.57. Finally the depth profile from the side view with the slope for $\dot{Q} = 0.0045 \text{ m}^3/\text{s}$, and the slopes are; $S = 0^\circ$, $S = 1.02^\circ$ and $S = 1.406^\circ$ is given in Fig. 3.58.

The velocity profile from the top view with a slope for each slope value and flow rate value is represented by Fig. 3.59, Fig. 3.60, and Fig. 3.61.

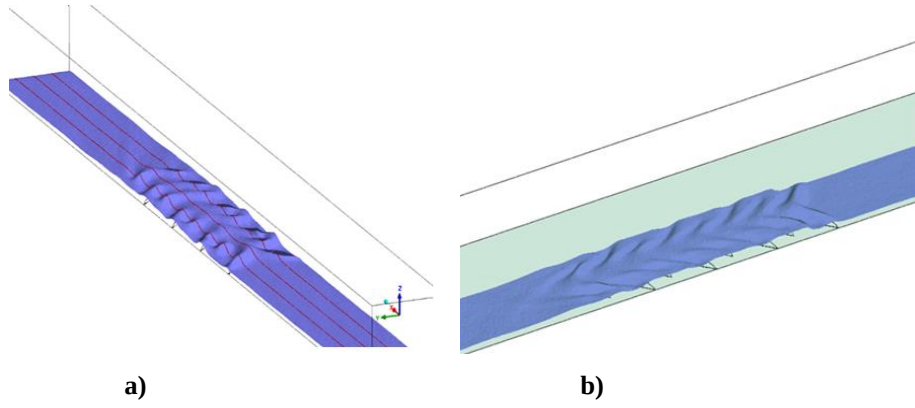


Figure 3.53. Top view flow at **a)** $S = 1.406^\circ$ with $\dot{Q} = 0.0035 \text{ m}^3/\text{s}$, **b)** $\dot{Q} = 0.0045 \text{ m}^3/\text{s}$



Figure 3.54. Water profiles from top view $S = 1.406^\circ$, $\dot{Q} = 0.0045 \text{ m}^3/\text{s}$

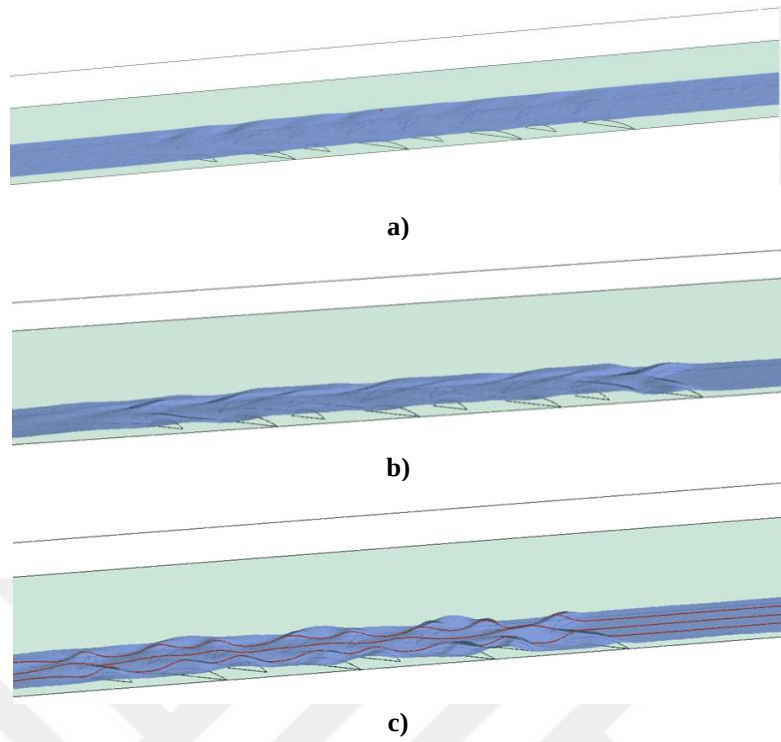


Figure 3.55. Depth profile from side view for $\dot{Q} = 0.0035 \text{ m}^3/\text{s}$, **a)** $S = 0^\circ$, **b)** $S = 1.0156^\circ$, **c)** $S = 1.406^\circ$



Figure 3.56. Depth profile in first obstacle explaining the flood jump, **a)** $S = 1.0156^\circ$, $\dot{Q} = 0.0035 \text{ m}^3/\text{s}$

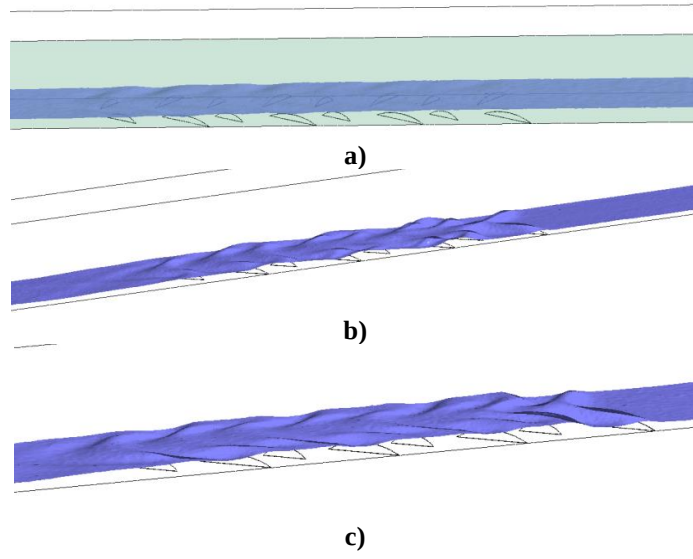


Figure 3.57. Water side profile at first obstacle, $S = 1.406^\circ$, $\dot{Q} = 0.004 \text{ m}^3/\text{s}$

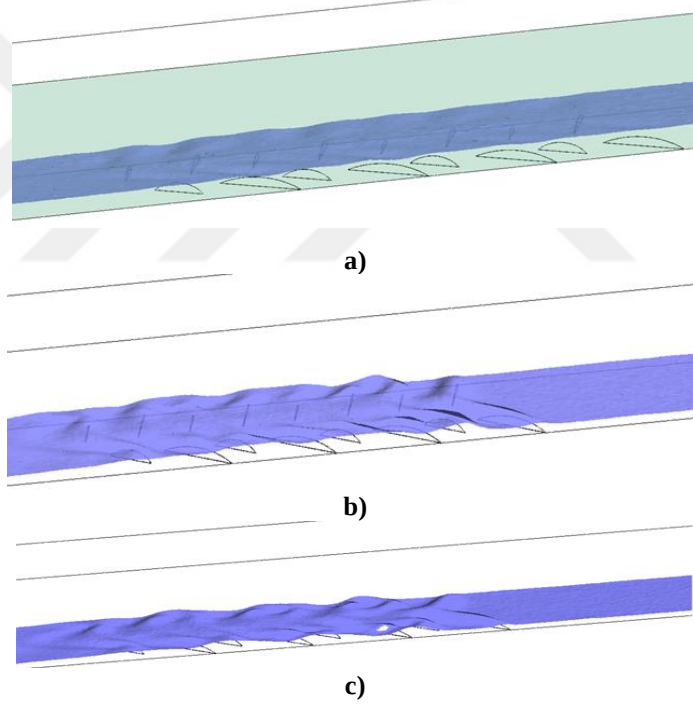


Figure 3.58. Water side profile at first obstacle, $S = 1.406^\circ$, $\dot{Q} = 0.0045 \text{ m}^3/\text{s}$

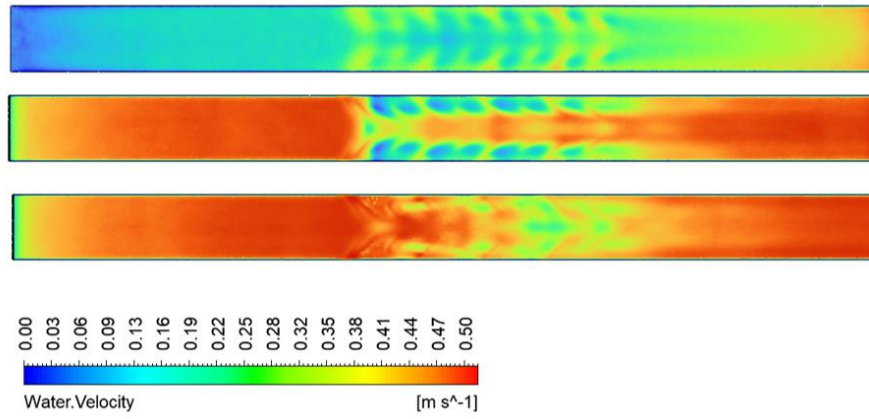


Figure 3.59. Velocity profile from top view for $\dot{Q} = 0.0035 \text{ m}^3/\text{s}$, **a)** $S = 0^\circ$, **b)** $S = 1.0156^\circ$, **c)** $S = 1.406^\circ$

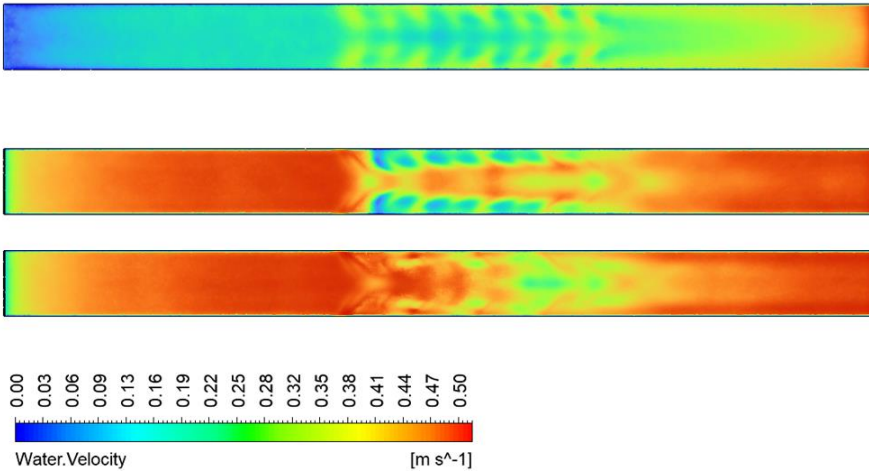


Figure 3.60. Velocity profile from top view for $\dot{Q} = 0.004 \text{ m}^3/\text{s}$, **a)** $S = 0^\circ$, **b)** $S = 1.0156^\circ$, **c)** $S = 1.406^\circ$

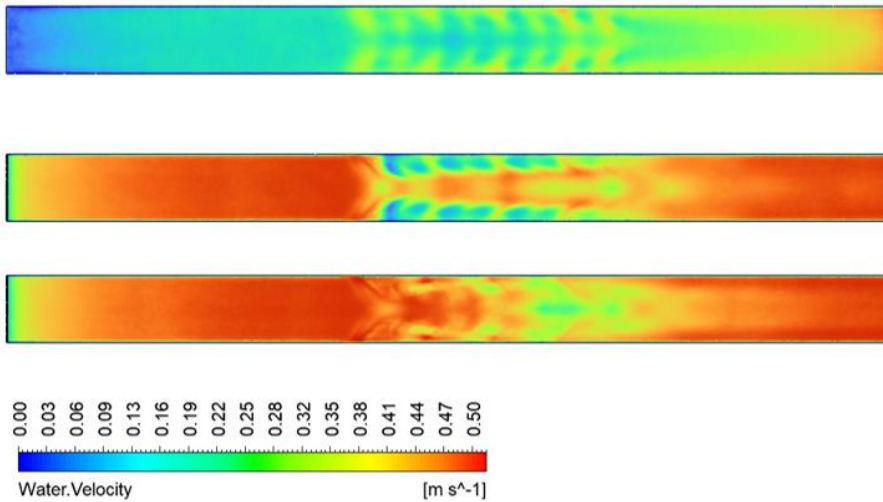


Figure 3.61. Velocity profile from top view for $\dot{Q} = 0.0045 \text{ m}^3/\text{s}$, **a)** $S = 0^\circ$, **b)** $S = 1.0156^\circ$, **c)** $S = 1.406^\circ$

3.3. Depth Analysis

3.3.1. Depth Analysis along Channel Distance with Rectangle Obstacles

Water depth along the channel for rectangular obstacles under the conditions of; $\dot{Q} = 0.0035 \text{ m}^3/\text{s}$, $S = 0^\circ$ is given in Fig. 3.62. The presence of obstacles in an open channel greatly affects various flow characteristics. As water flows through the channel, irregularities caused by obstacles change the behavior of the flow depth. These changes propagate downstream, affecting phenomena such as separation and eddy formation. Obstacles create disturbances in the flow pattern, leading to break zones where the flow separates from the obstruction surfaces. This separation changes the flow behavior, causing fluctuations in flow depth along the channel. In addition, irregularities caused by obstacles can lead to the formation of vortices downstream, contributing to mixing and energy dissipation within the flow. Furthermore, obstacles exert drag forces on flowing water, which vary depending on factors such as the shape, size, and direction of the obstacle. These drag forces affect the flow depth distribution and contribute to the overall energy dissipation in the system.

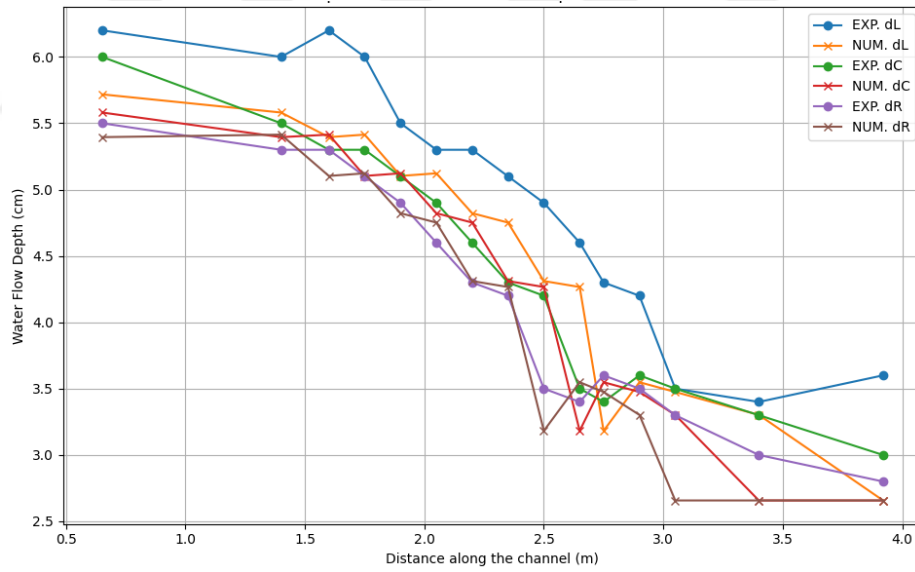


Figure 3.62. Water depth along channel for rectangular obstacles; $\dot{Q} = 0.0035 \text{ m}^3/\text{s}$, $S = 0^\circ$

Variations in flow depth caused by obstructions also lead to changes in shear stress along the channel bottom. Higher flow depths correspond to increased shear stress, resulting in enhanced bed shear and sediment transport. To evaluate the accuracy of the numerical simulations compared to the experimental results, the study has been calculated the percentage error for each data point. The percentage error is calculated as the absolute difference between the experimental and numerical depth divided by the experimental depth multiplied by 100%. Lower

percentage errors indicate better agreement between experimental and numerical data. The average percentage error for the left side of the channel is 5.5572%, for the center is 3.6823%, and for the right side is 6.2846%. These values provide insight into the agreement between experimental and numerical data and help identify areas where improvements in modeling or experimental techniques may be necessary.

In the case of $\dot{Q} = 0.004 \text{ m}^3/\text{s}$, $S = 0^\circ$, the dimensions of the channel remain constant at 400 cm in length, 30 cm in width, and 40 cm in height (Fig. 3.63). Experimental and numerical data for this flow rate reveal variations in water depth along the channel, with experimental data ranging from 2.6 cm to 5.5 cm for both the left (d_l) and center depths (d_c), and the right depth (d_r) showing a similar range. The corresponding numerical data also show variations in depth along the channel, with values ranging from 2.35 cm to 5.51 cm for all sections. The percentage errors between the experimental and numerical results were calculated, demonstrating the accuracy of the numerical model compared to experimental measurements. The average percentage of errors for each section (left, center, right) is 3.5% for the left, 1.43% for the center, and 0.95% for the right. These errors indicate strong agreement between experimental and numerical results, with relatively low discrepancies.

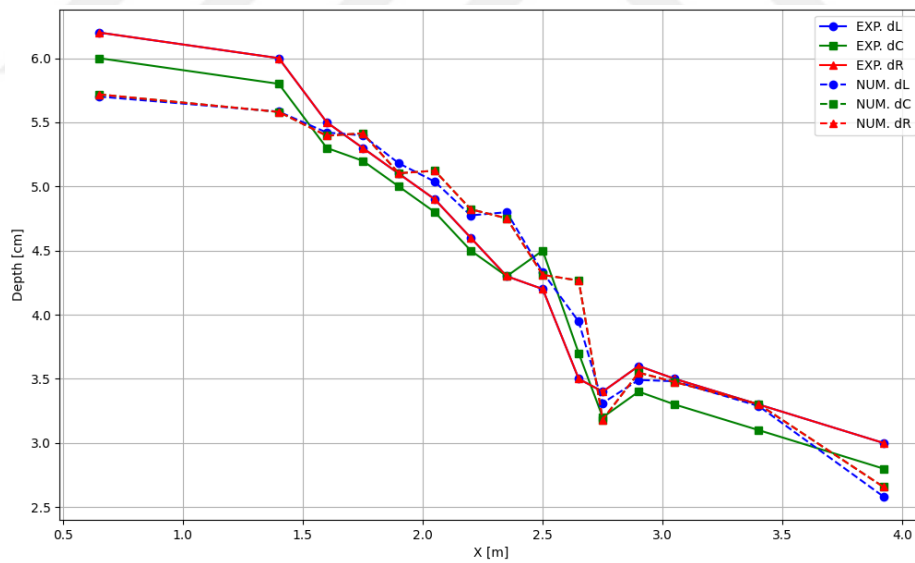


Figure 3.63. Water depth along channel for rectangular obstacles; $\dot{Q} = 0.004 \text{ m}^3/\text{s}$, $S = 0^\circ$

The presence of obstructions affects the depth of water flow in several ways. Disturbances in the flow pattern create separation zones where the flow deviates from obstacles. In addition, the irregular geometry of obstacles creates vortices and turbulence, which facilitate energy dissipation and mixing within the flow. Interaction with flowing water creates obstacles to the drag force, affecting its stability and movement. Moreover, the presence of obstacles changes the

flow velocity gradient near the channel bottom, resulting in varying shear stress distribution along the channel bottom. In conclusion, the experimental and numerical results provide valuable insights into the complex behavior of water flow in channels with obstructions. Understanding this behavior is important across applications such as hydraulic engineering, river management, and environmental studies. The agreement between experimental and numerical data confirms the effectiveness of numerical modeling in predicting flow behavior and its interaction with obstacles in open channels.

In the case of $\dot{Q} = 0.0045 \text{ m}^3/\text{s}$, $S = 0^\circ$, the analysis of the experimental data reveals fluctuations in depth across the left, center, and right sections, ranging from 2.6 cm to 5.7 cm, 2.2 cm to 5.9 cm, and 2.8 cm to 5.7 cm, respectively as shown in Fig. 3.64. Similarly, numerical simulations capture depth variations spanning from 2.35 cm to 5.83 cm (left), 2.17 cm to 5.84 cm (center), and 2.57 cm to 5.79 cm (right). Calculation of the percentage error between experimental and numerical results yields the average percentage error for the left, center, and right sections to be approximately 7.8%, 12.7%, and 7.9%, respectively. These values indicate a significant deviation between the datasets, with notably higher errors observed in the center section compared to the left and right sections. Despite the discrepancies, the presence of obstacles continues to exert a pronounced influence on flow behavior, contributing to flow separation, eddies formation, and alterations in shear stress distribution.

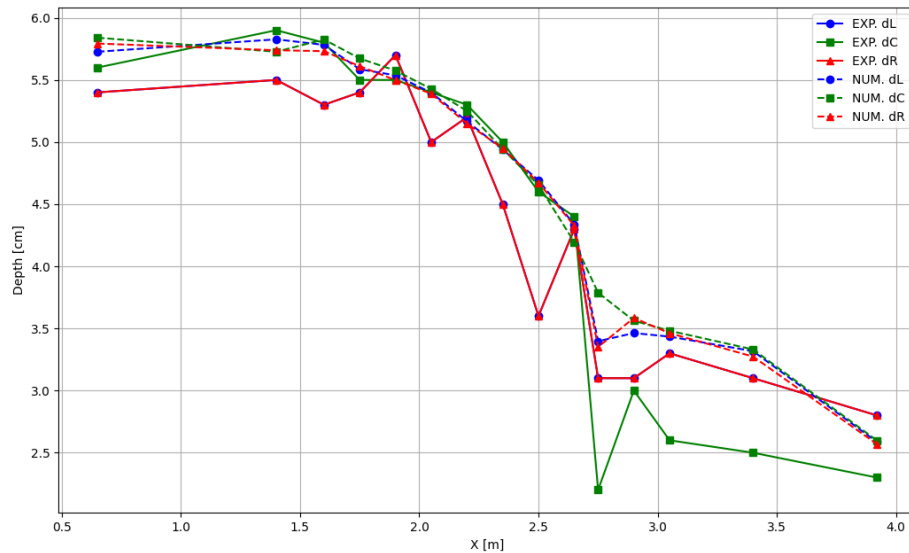


Figure 3.64. Water depth along channel for rectangular obstacles; $\dot{Q} = 0.0045 \text{ m}^3/\text{s}$, $S = 0^\circ$

As it is seen in Fig. 3.65 ($\dot{Q} = 0.0035 \text{ m}^3/\text{s}$, $S = 1.02^\circ$) experimental observations range from 1.3 cm to 4.2 cm (left), from 1.7 cm to 4.3 cm (center), and 1.5 cm to 4.1 cm (right). In

contrast, numerical simulations capture depth fluctuations spanning 1.56 cm to 4.17 cm (left), 1.59 cm to 4.15 cm (center), and 1.51 cm to 4.13 cm (right).

Average errors for the left, center, and right sections are about 20.70 %, 14.92 %, and 20.78 %, respectively. These values indicate a relatively low degree of skewness between the data sets, especially in the left and right sections. The presence of obstacles significantly affects flow dynamics, contributing to variations in flow depth and inducing complex flow patterns characterized by flow separation and vortex formation. Furthermore, obstacles experience drag forces, which affect their stability and mobility. These insights underscore the importance of accurate modeling of flow behavior in obstructed channels, providing valuable implications for hydraulic engineering, river management, and environmental studies.

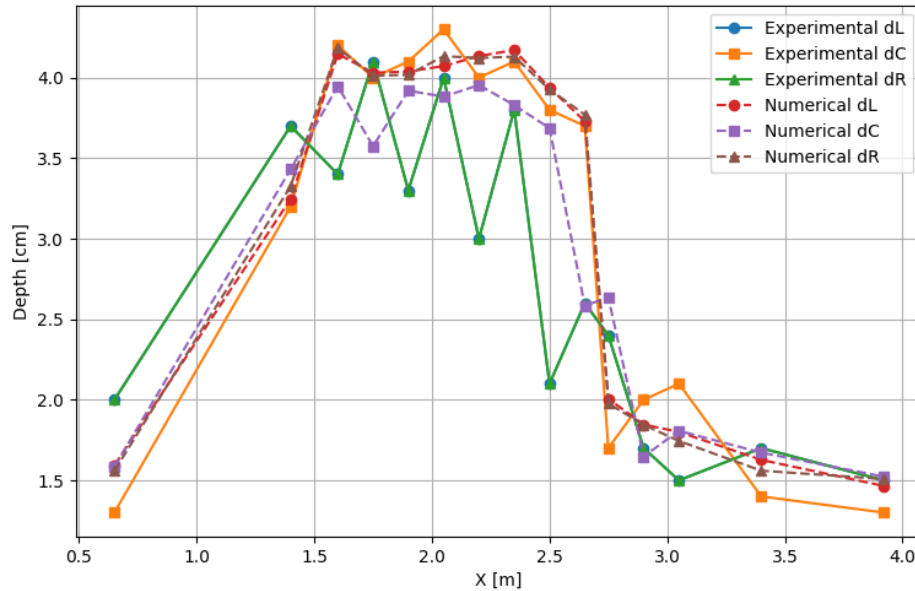


Figure 3.65. Water depth along channel for rectangular obstacles; $\dot{Q} = 0.0035 \text{ m}^3/\text{s}$, $S = 1.0156^\circ$

Figure 3.66 shows that increasing the flow rate to $0.004 \text{ m}^3/\text{s}$ provides deeper insights into the flow behavior under different hydraulic conditions. Experimental observations indicate variations of 1.4 cm to 4.6 cm (left), 1.45 cm to 4.3 cm (center), and 1.5 cm to 4.2 cm (right). In contrast, numerical simulations capture depth fluctuations spanning from 1.76 cm to 4.52 cm (left), 1.76 cm to 4.33 cm (center), and 1.62 cm to 4.32 cm (right). Average errors for the left, center, and right sections are about 15.9%, 1.4%, and 15.9%, respectively.

According to Fig. 3.67, increasing the flow rate to $0.0045 \text{ m}^3/\text{s}$, while maintaining the same slope angle of 1.02° results in noticeable changes in the depth of water flow. Similar to previous cases, experimental and numerical data sets reveal fluctuations in water depth across the left, center, and right sections of the channel. Experimental observations indicate variations of 1.8 cm to 5.3 cm (left), 1.7 cm to 4.7 cm (center), and 1.65 cm to 4.4 cm (right). In contrast, numerical

simulations capture depth fluctuations spanning from 1.96 cm to 4.67 cm (left), 1.97 cm to 4.77 cm (center), and 1.76 cm to 4.57 cm (right). By calculating the error percentages between the experimental and numerical results, the average error percentage for the left, center, and right sections were about 1.1%, 3.5%, and 1.9%, respectively.

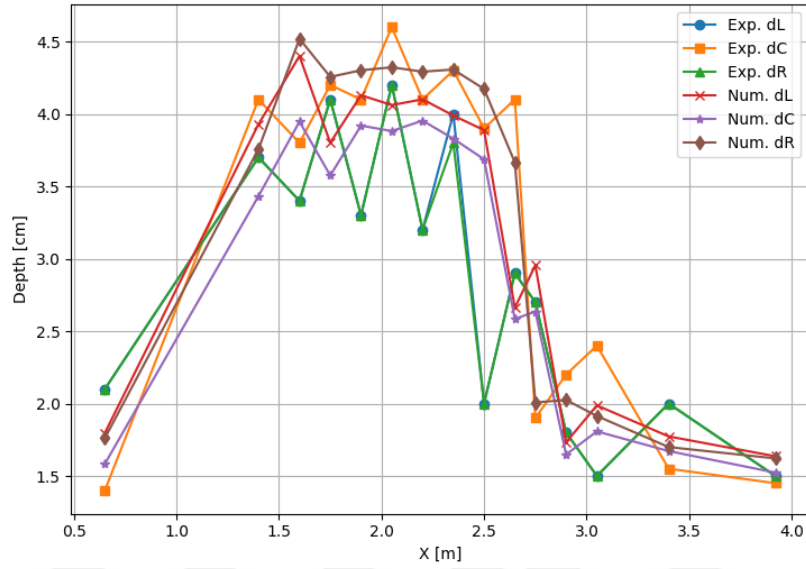


Figure 3.66. Water depth along channel for rectangular obstacles; $\dot{Q} = 0.004 \text{ m}^3/\text{s}$, $S = 1.0156^\circ$

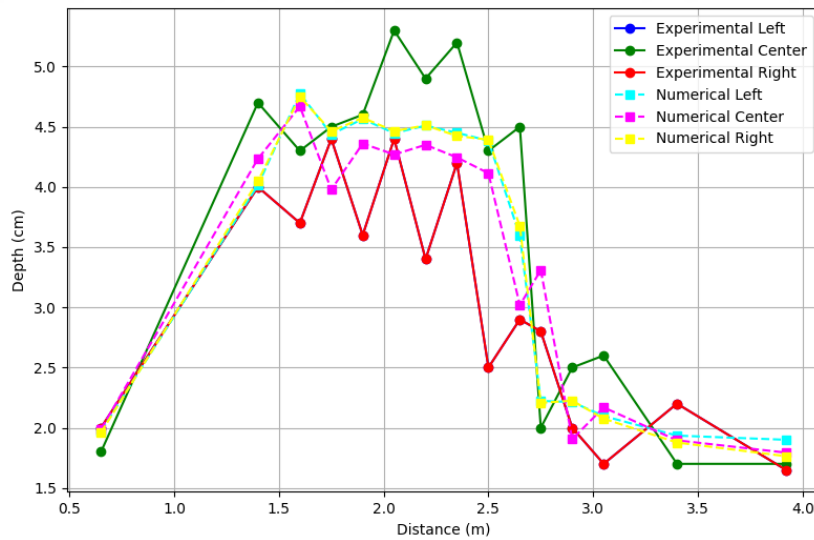


Figure 3.67. Water depth along channel for rectangular obstacles; $\dot{Q} = 0.0045 \text{ m}^3/\text{s}$, $S = 1.0156^\circ$

For the case of $\dot{Q} = 0.0035 \text{ m}^3/\text{s}$, $S = 1.406^\circ$ Fig. 3.68 is drawn. It shows that experimental observations revealed fluctuations in water depth, ranging from 1.3 cm to 4.6 cm (left), 1.45 cm to 4.3 cm (center), and 1.5 cm to 4.2 cm (right). In contrast, numerical simulations depicted

slightly different depth ranges, extending from 1.56 cm to 4.27 cm (left), 1.36 cm to 4.27 cm (center), and 1.29 cm to 4.29 cm (right). The average errors for the left, center, and right sections are about 16.9%, 2.9%, and 16.8%, respectively.

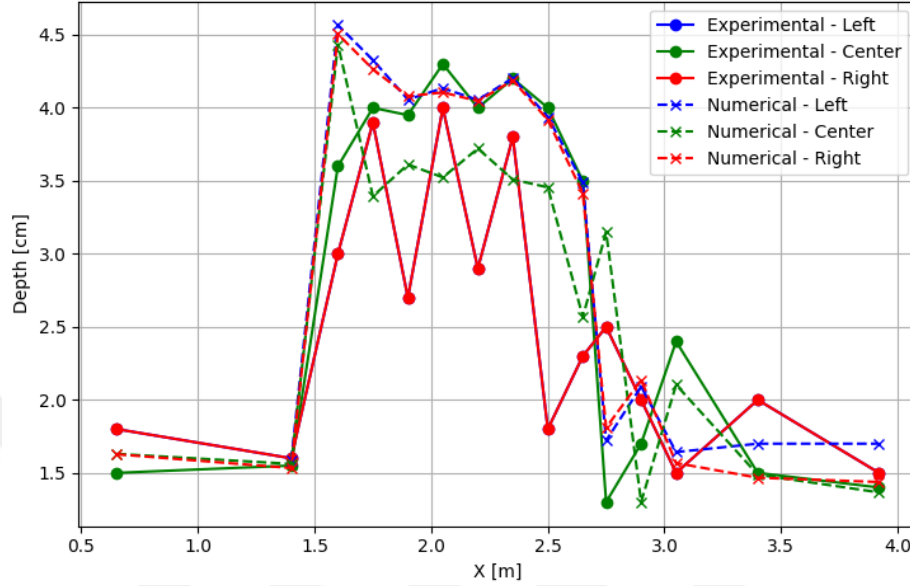


Figure 3.68. Water depth along channel for rectangular obstacles; $\dot{Q} = 0.0035 \text{ m}^3/\text{s}$, $S = 1.406^\circ$

In the case of $\dot{Q} = 0.004 \text{ m}^3/\text{s}$, $S = 1.406^\circ$, Fig. 3.69 is developed. The comparison between experimental and numerical results across various locations along the channel sheds light on how obstacles influence water flow depth and associated flow characteristics. At the distance; $X = 0.6509 \text{ m}$, where the obstacle is first encountered, there's a notable disparity between experimental and numerical data, indicating a significant impact of obstacles on water depth. This discrepancy suggests that the presence of obstacles leads to flow separation, eddies formation, and alterations in drag force and shear stress. As the flow progresses downstream, the influence of obstacles varies.

At $X=1.4 \text{ m}$ and $X=2.65 \text{ m}$, the experimental and numerical results align more closely, suggesting a moderate influence of obstacles. However, at $X=1.75 \text{ m}$ and $X=2.05 \text{ m}$, significant discrepancies between experimental and numerical data are observed, particularly for d_l , d_c , and d_r indicating a substantial impact of obstacles on flow patterns. The discrepancies in water depth between experimental and numerical results imply variations in flow behavior induced by obstacles. These variations could lead to the formation of eddies and separation zones, affecting drag force and shear stress distribution along the channel. The observed error percentages further highlight the challenges in accurately modeling flow dynamics in the presence of obstacles. The highest average percentage error for d_l (20.13%) suggests significant discrepancies in the left side of the channel, potentially indicating enhanced separation and eddies formation due to obstacles.

Similarly, the high error percentage for d_r (14.35%) signifies alterations in flow patterns and drag force distribution on the right side. In contrast, the lowest average percentage error for d_c (0.64%) suggests relatively better agreement between experimental and numerical data in the central region, indicating a lesser impact of obstacles.

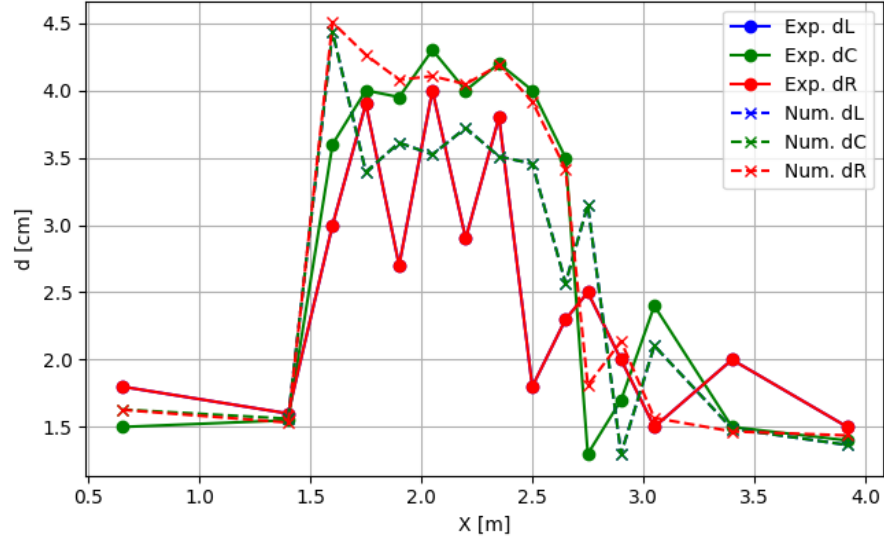


Figure 3.69. Water depth along channel for rectangular obstacles; $\dot{Q} = 0.004 \text{ m}^3/\text{s}$, $S = 1.406^\circ$

Figure 3.70 shows the case of $\dot{Q} = 0.0045 \text{ m}^3/\text{s}$, $S = 1.406^\circ$. The average errors of d_l , d_c , and d_r are found 13.41%, 4.26%, and 14.37% respectively, highlighting the reliability and accuracy of both experimental measurements and numerical simulations.

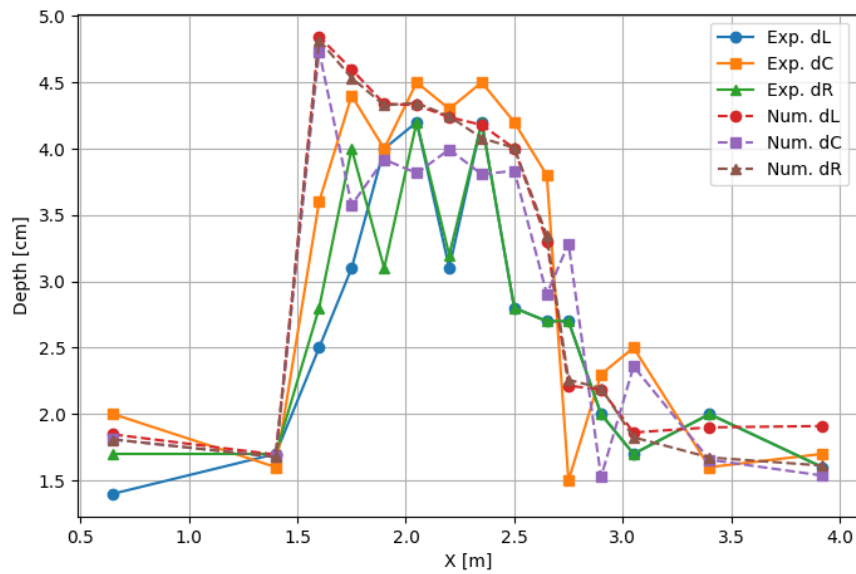


Figure 3.70. Water depth along channel for rectangular obstacles; $\dot{Q} = 0.0045 \text{ m}^3/\text{s}$, $S = 1.406^\circ$

Some numerical results of depths analysis will be mentioned here. Fig. 3.71 represents a summary of all rectangle obstacles cases by means of depth analysis. All of them are explained above with details.

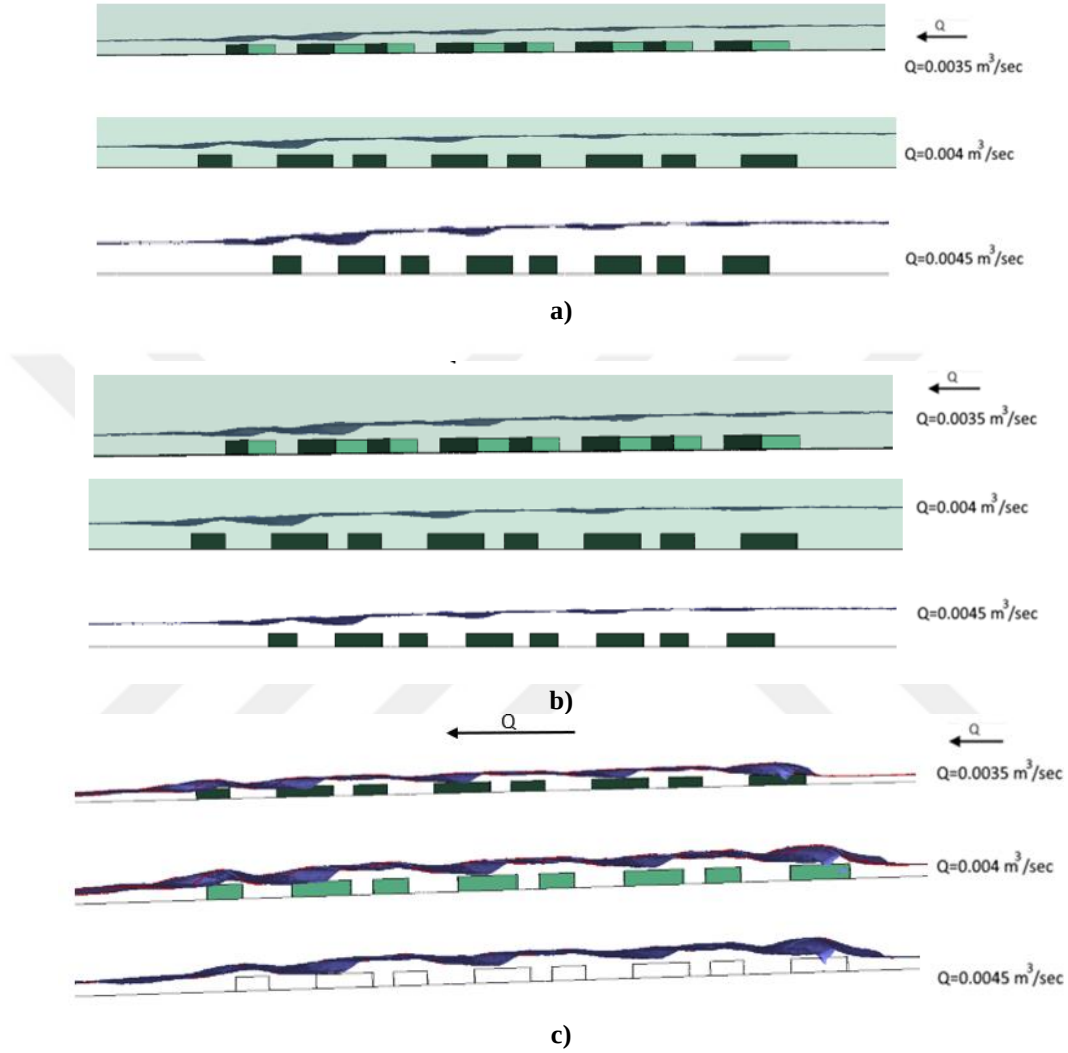


Figure 3.71. Numerically comparison of all depth values **a)** $S = 0^\circ$, **b)** $S = 1.0156^\circ$, **c)** $S = 1.406^\circ$

3.3.2. Depth Analysis along Channel Distance with Rectangular Arc Obstacle

For this tested case ($\dot{Q} = 0.0035 \text{ m}^3/\text{s}$, $S = 0^\circ$) The average errors for d_l , d_c , and d_r are calculated as 8.6%, 2.9%, and 8.5%, respectively. Obstructions such as rectangular arches disrupt water flow, causing changes in water flow depth. As can be seen from experimental and numerical data plotted in Fig. 3.72, there are differences between the predicted and actual water flow depths.

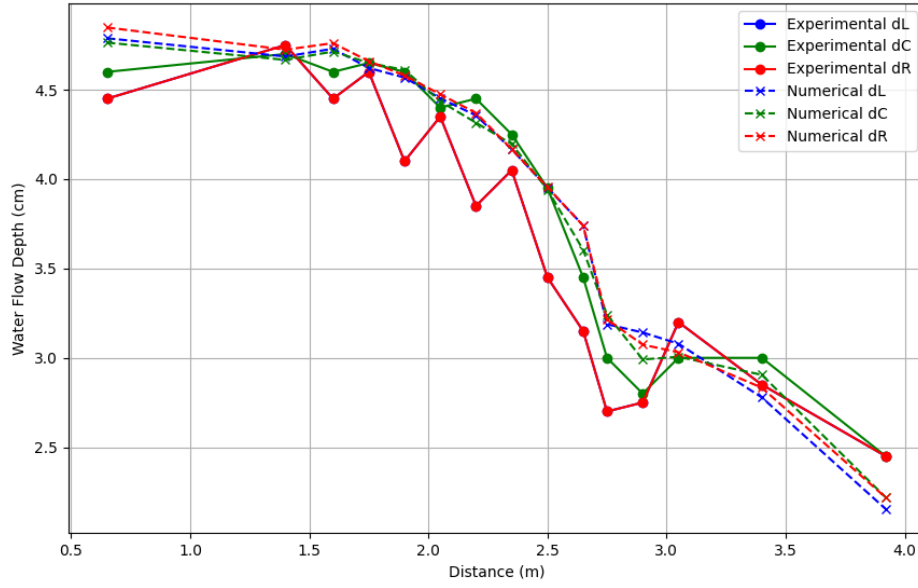


Figure 3.72. Water depth along channel for rectangular arc obstacles; $\dot{Q} = 0.0035 \text{ m}^3/\text{s}$, $S = 0^\circ$

Average errors indicate the discrepancy between predicted and actual values, indicating that obstructions affect the depth of water flow. Obstructions cause disturbances in the flow, leading to the formation of eddies. Whirlpools are swirling eddies or currents that form when water flow encounters an obstruction. The presence of eddies changes flow patterns and can lead to variations in water depth and velocity around the obstruction. Obstructions exert a drag force on flowing water. This drag force results from the interaction between the obstacle and the moving fluid. The magnitude of the drag force depends on factors such as the shape and size of the obstacle, as well as the velocity of the flowing water. Rectangular bow baffles, with their specific shape, contribute to the overall drag force experienced by the water flow. Shear stress refers to the force per unit area acting parallel to the surface of the obstacle. When water flows through an obstacle, shear stress is generated along the boundary between the flowing water and the surface of the obstacle. The presence of rectangular arc obstructions changes the distribution of shear stress along the water surface, affecting sediment transport and erosion patterns. Obstructions can cause flow separation, where flowing water separates from the surface of the obstruction. This separation occurs due to differences in pressure and velocity around the obstacle. In the case of obstructions with a rectangular arch, flow separation may lead to the formation of recirculation zones behind the obstruction, affecting the overall flow dynamics.

The curves shown in Fig. 3.73 for the case of $\dot{Q} = 0.004 \text{ m}^3/\text{s}$, $S = 0^\circ$ show experimental and numerical values of water depth d_d for different distances X along the channel. From the curves, in study can note that both experimental and numerical results generally follow a similar

trend, showing a decrease in water depth with increasing distance X . However, there are some differences between experimental and numerical values at certain points along the channel.

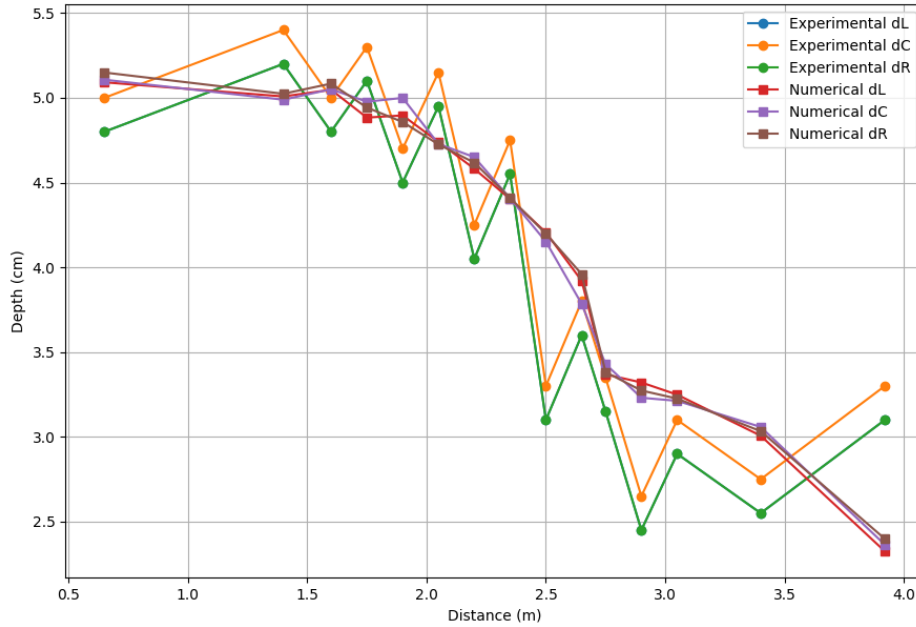


Figure 3.73. Water depth along channel for rectangular arc obstacles; $\dot{Q} = 0.004 \text{ m}^3/\text{s}$, $S = 0^\circ$

The average of d_l , d_c , and d_r errors is calculated as 12.7%, 9.5% and respectively 12.6%. These errors indicate discrepancies between experimental and numerical values. A higher error rate indicates a greater deviation between the two data sets. Discrepancies between experimental and numerical results can be attributed to various factors such as inaccuracies in measurement, assumptions made in numerical simulations, and complexities in flow behavior around obstacles. These deviations can have significant effects on water flow behavior, including differences in separation, vortex formation, drag force, and shear stress. Higher errors may indicate areas where flow behavior is more complex or where the numerical model may need improvement to better capture physical phenomena. Understanding these discrepancies is crucial to accurately predict and model flow behavior in practical applications, such as hydraulic engineering projects or environmental assessments. In conclusion, while both experimental and numerical approaches provide valuable insights into water flow behavior, it is necessary to carefully analyze and reconcile any differences between the two data sets to ensure accurate predictions and informed decisions in real-world scenarios.

According to Fig. 3.74, namely for $\dot{Q} = 0.0045 \text{ m}^3/\text{s}$, $S = 0^\circ$, the average percentage errors for d_l , d_c , and d_r are 9.689%, 7.278%, and 12.674%, respectively. These values indicate the average deviation of the experimental results from the numerical (or expected) values. Looking at the experimental data, depth measurements (d_l , d_c , and d_r) has been observed variations in water

flow at different distances X along the channel. The discrepancies between the experimental and numerical values could be attributed to various factors such as measurement errors, experimental conditions, and uncertainties in the flow dynamics. When comparing the experimental and numerical values at each distance X , fluctuations in the water flow depths have been noted. These fluctuations suggest that the channel geometry, flow velocity, and other hydraulic parameters influence the distribution of water depths along the channel. The discrepancies in water flow depth measurements can lead to differences in flow characteristics such as eddy formation, drag force, and shear stress. These variations can affect the overall flow behavior and its interaction with obstacles in the channel. It's essential to consider the sources of error and uncertainties in experimental setups to improve the accuracy of water flow depth measurements.

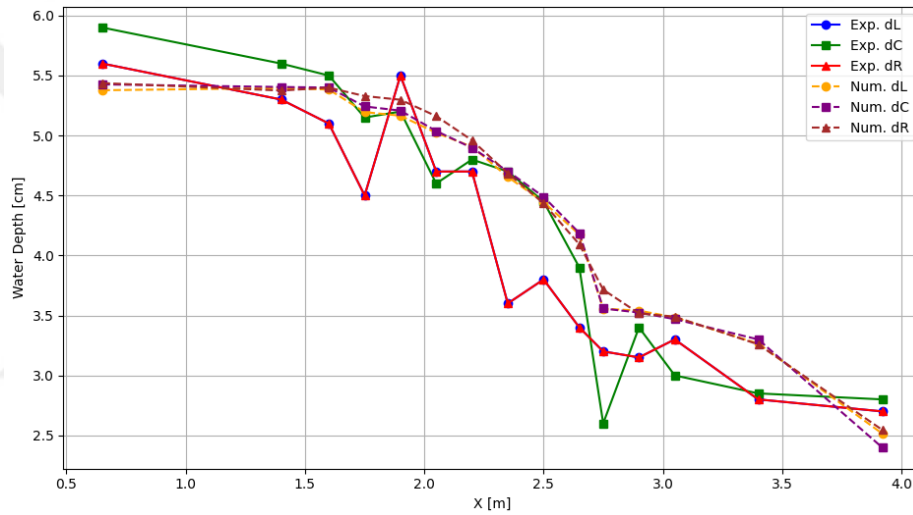


Figure 3.74. Water depth along channel for rectangular arc obstacles; $\dot{Q} = 0.0045 \text{ m}^3/\text{s}$, $S = 0^\circ$

In Fig. 3.75, when $\dot{Q} = 0.0035 \text{ m}^3/\text{s}$, $S = 1.02^\circ$, due to the results of average percentage error (d_l : 31.2%, d_c : 9.1% and d_r : 32.3%), the observed discrepancies between experimental and numerical data for water flow depth (d_a) can be attributed to various factors, including the presence of obstructions in the flow. The presence of obstacles, such as those encountered in experiments, can significantly affect the behavior and values of water flow depth (d_a). These obstructions create disturbances in the flow field, resulting in complex flow patterns characterized by changes in velocity, pressure, and turbulence intensity. As a result, the measured water depths may deviate from the ideal conditions assumed in numerical simulations, leading to discrepancies between experimental and numerical data. One of the primary effects of obstructions is the generation of flow separation zones. When the flow encounters an obstacle, it is forced to deviate from its original path, creating separation zones where the flow separates from the surface of the obstacle. This can lead to local variations in water depth, especially below obstacles, where the

flow returns to the surface. In addition, obstructions can induce the formation of eddies and vortices in the flow. These vortex movements redistribute momentum and energy within the flow field, causing fluctuations in water depth along the flow direction. The presence of eddies and vortices can amplify turbulence levels and change flow characteristics, further complicating water depth prediction using numerical models. Furthermore, obstructions can increase the drag force and shear stress generated by flowing water. When water interacts with the surface of an obstacle, it encounters resistance, resulting in energy loss and changes in flow velocity. This can affect the distribution of water depth profiles, especially in areas close to obstacles, where flow velocity is greatly affected by drag force. Overall, the effects of obstructions on water flow depth are multifaceted and complex. While numerical models strive to capture these complexities, inherent uncertainties associated with experimental setups and numerical simulations can lead to discrepancies between observed and simulated data.

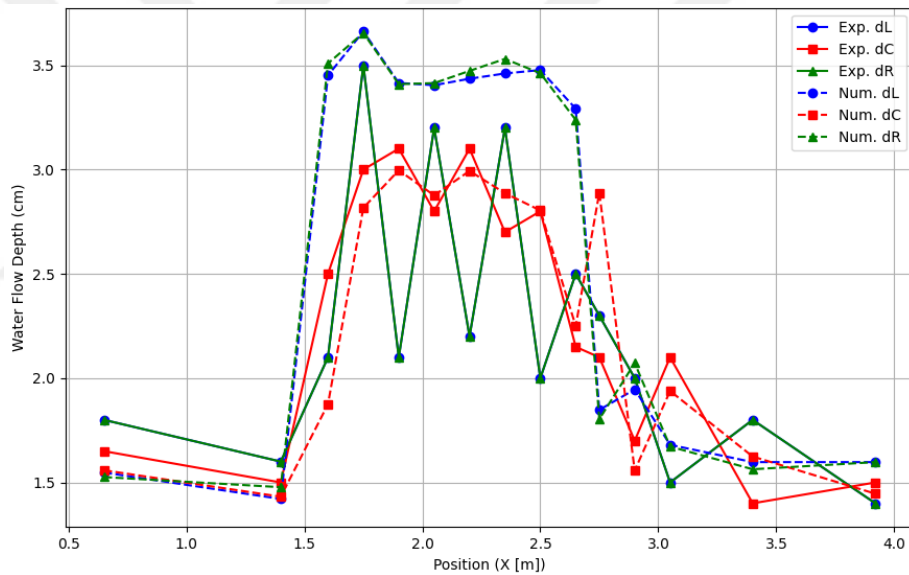


Figure 3.75. Water depth along channel for rectangular arc obstacles; $\dot{Q} = 0.0035 \text{ m}^3/\text{s}$, $S = 1.0156^\circ$

In the case of $\dot{Q} = 0.004 \text{ m}^3/\text{s}$, $S = 1.02^\circ$, as shown in Fig. 3.76, comparison of experimental and numerical data for water flow depth (d_l , d_c , and d_r) along different locations in the channel reveals several noteworthy trends and variations. First, discrepancies between experimental and numerical results appear in all parameters and situations. These differences likely stem from factors such as measurement inaccuracy, differences in flow conditions, and the inherent complexity of digital hydraulic models. By examining the data, fluctuations in the depth of water flow with changes in locations along the channel (X) strongly noted. Remarkably, at $X = 1.6 \text{ m}$, The average percentage errors of d_l , d_c , and d_r are relatively high: about 22.27%, 18.67%, and 22.25%, respectively. These numbers confirm that there are large discrepancies between

experimental and numerical results, indicating limitations in the accuracy of the numerical model or potential problems in the experimental setup. Moreover, increasing the flow rate to $0.0045 \text{ m}^3/\text{s}$ can exacerbate the discrepancies between experimental and numerical results, because it may change the flow patterns and hydraulic behavior.

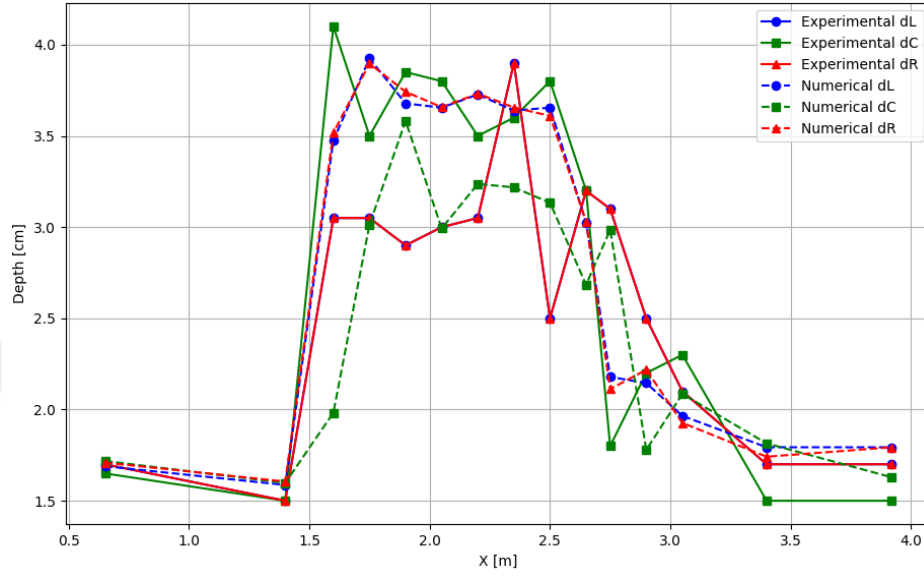


Figure 3.76. Water depth along channel for rectangular arc obstacles; $\dot{Q} = 0.004 \text{ m}^3/\text{s}$, $S = 1.0156^\circ$

For $\dot{Q} = 0.0045 \text{ m}^3/\text{s}$, $S = 1.02^\circ$, when examining the experimental and numerical results of water depth measurements over different distances (X), it is clear that there are differences between the two data sets. As is seen in Fig. 3.77, while the general trend in water depth variation concerning distance appears consistent between experimental and numerical data, there are notable differences in the exact depth values at specific distances. Experimental data, derived from direct measurements in a physical experiment setting, provide insight into actual water depth values. On the other hand, numerical data, obtained through computational models based on fluid dynamics equations, provide simulated values of water depth. These discrepancies become apparent when comparing the two data sets. For example, at certain distances such as $X = 1.6\text{m}$, the experimental depth values of d_l and d_c are significantly lower compared to the corresponding numerical values. On the contrary, at $X = 2.2\text{m}$, the experimental depth values of the d_c exceed the numerical values. The average percentage errors, calculated as the average deviation between the experimental and numerical results, shed light on the overall accuracy of the numerical model compared to the experimental data (d_l : 15.1%, d_c : 16.63%, and d_r : 15.69%).

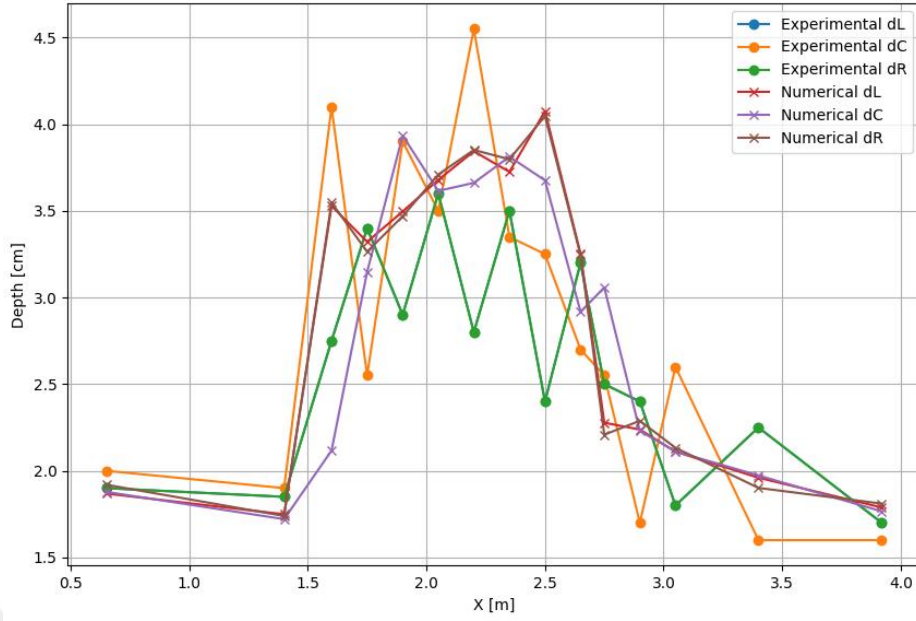


Figure 3.77. Water depth along channel for rectangular arc obstacles; $\dot{Q} = 0.0045 \text{ m}^3/\text{s}$, $S = 1.0156^\circ$

In the case of $\dot{Q} = 0.0035 \text{ m}^3/\text{s}$, $S = 1.406^\circ$, comparing the experimental and numerical results as seen in Fig. 3.78, differences appear in the water depth values at different distances. For example, at $X = 1.6 \text{ m}$, the experimental depth values of d_l and d_r significantly exceed the corresponding numerical values. On the contrary, at $X = 2.2 \text{ m}$, the experimental depth values of the d_c exceed the numerical values. The average percentage errors of d_l , d_c , and d_r calculated as the average deviation between the experimental and numerical results, highlight the overall accuracy of the numerical model relative to the experimental data. These errors underscore the extent of the discrepancy between the two data sets. Specifically, the average percentage error of the deciliter is about 33.30%, which means that there is a large deviation between the experimental and numerical results for this parameter. Likewise, the average percentage errors for d_c and d_r are approximately 12.61% and 33.25%, respectively, indicating varying degrees of accuracy for these parameters.

In the case of $\dot{Q} = 0.004 \text{ m}^3/\text{s}$, $S = 1.406^\circ$, it can be said that upon inspection, it is clear that there are differences between experimental and numerical measurements for all three parameters d_l , d_c , and d_r . These differences indicate differences in the precision or accuracy of the experimental setup and the numerical model. It is all shown in Fig. 3.79. For example, at $X = 1.6 \text{ m}$, the experimental depth measurements of d_l and d_r are significantly lower compared to the numerical values, while the experimental d_c measurement exceeds the numerical value. Likewise, at $X = 2.35 \text{ m}$, the experimental depth measurements of d_l and d_r are higher than the corresponding numerical values. Moreover, by analyzing the average percentage errors for d_l , d_c ,

and d_r the average error per deciliter found by: 24.43%, an average d_c error rate: of 23.48%, average error in d_l : of 21.76%.

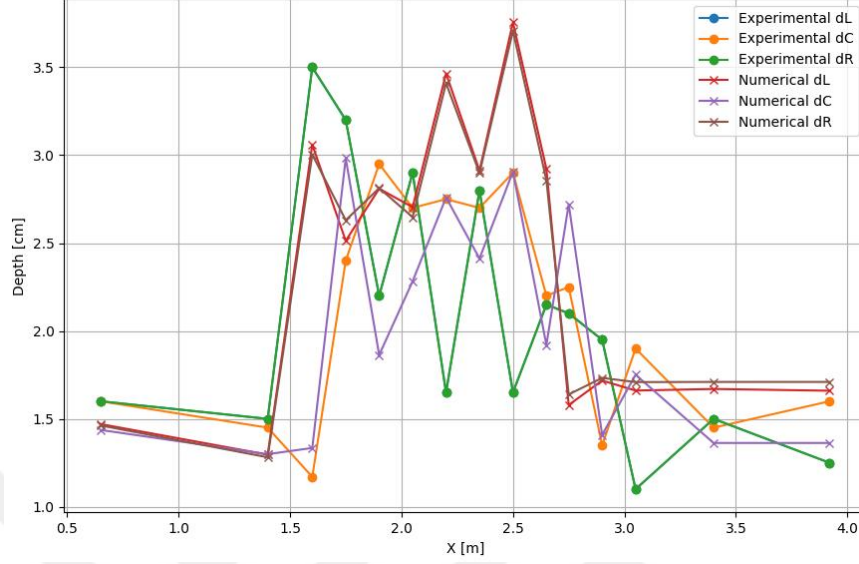


Figure 3.78. Water depth along channel for rectangular arc obstacles; $\dot{Q} = 0.0035 \text{ m}^3/\text{s}$, $S = 1.406^\circ$

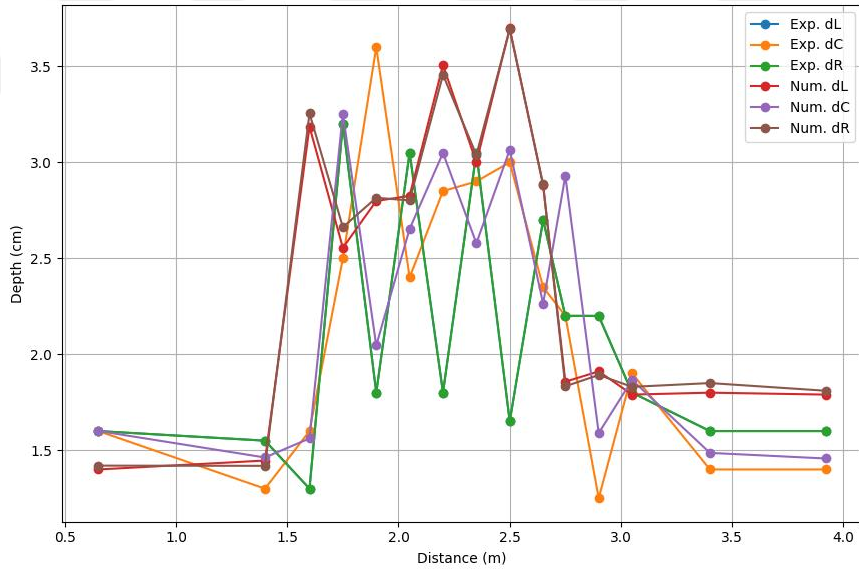


Figure 3.79. Water depth along channel for rectangular arc obstacles; $\dot{Q} = 0.004 \text{ m}^3/\text{s}$, $S = 1.406^\circ$

$\dot{Q} = 0.004 \text{ m}^3/\text{s}$, $S = 1.406^\circ$ case of rectangular arc obstacles is represented in Fig. 3.80. These differences are measured by the average percentage error for each parameter: d_l , d_c and d_r . For deciliter, the average percentage error is about 32.70%, indicating a large deviation between the experimental and numerical values. This indicates that the numerical model may not accurately capture the behavior of d_l over different distances. Likewise, for d_c , the average

percentage error is about 11.13%. While this error is relatively smaller than the deciliter error, it still indicates a noticeable difference between the experimental and numerical results. For d_r , the average percentage error is about 33.06%, reflecting the observed trend for d_l . This indicates that there is a large discrepancy between the experimental and numerical values of d_r over different distances.

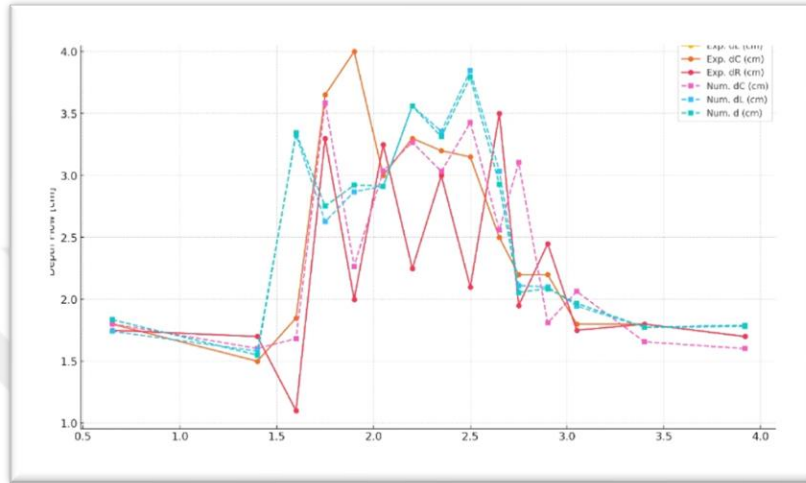


Figure 3.80. Water depth along channel for rectangular arc obstacles; $\dot{Q} = 0.0045 \text{ m}^3/\text{s}$, $S = 1.406^\circ$

Some numerical results of depth analysis will be given below. Fig. 3.81, Fig.3.82, and Fig. 3.83 respectively show the depths for the rectangular arc obstacle, from the side view points, respectively for each flowrate value.

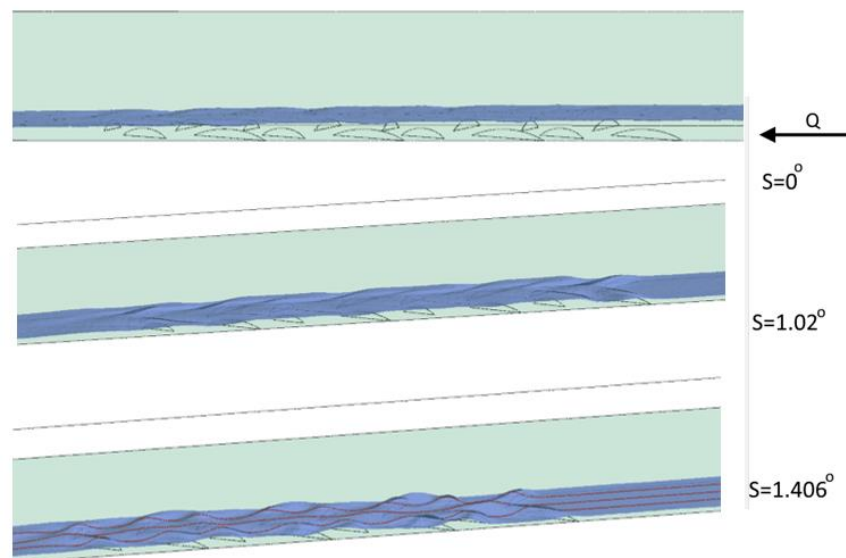


Figure 3.81. Depth for the rectangular arc obstacle, from the side view for $\dot{Q} = 0.0035 \text{ m}^3/\text{s}$

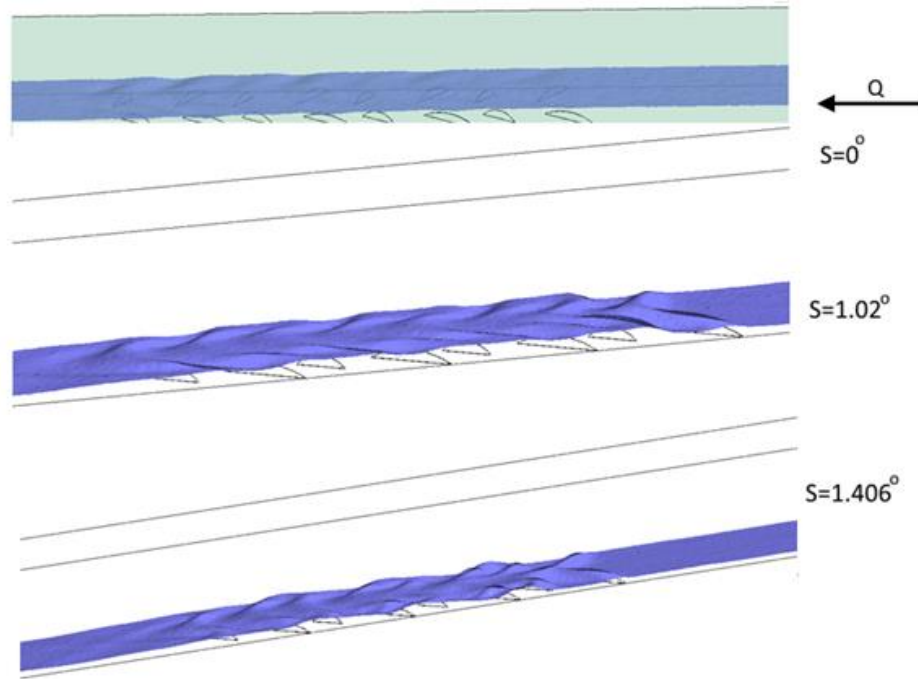


Figure 3.82. Depth for the rectangular arc obstacle, from the side view for $\dot{Q} = 0.004 \text{ m}^3/\text{s}$

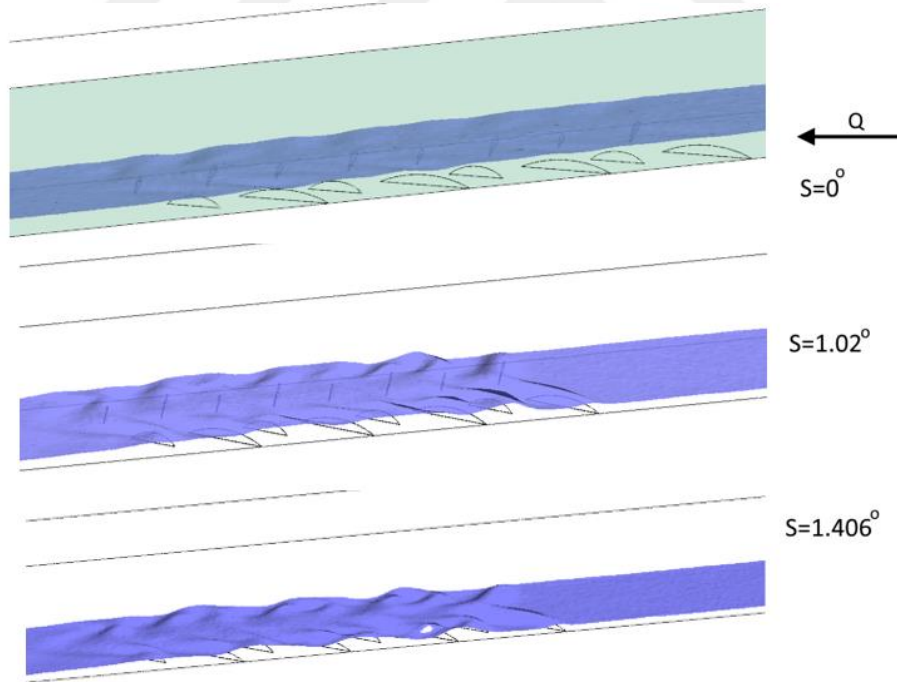


Figure 3.83. Depth for the rectangular arc obstacle, from the side view for $\dot{Q} = 0.0045 \text{ m}^3/\text{s}$

3.3.3. Depth Analysis along Channel Distance with Cube Obstacles

For the cube obstacles, when $\dot{Q} = 0.0035 \text{ m}^3/\text{s}$, $S = 0^\circ$, the average errors highlighted the accuracy of the numerical simulation in predicting water flow depths as seen in Fig. 3.84. For decimeters, the average error is 4.72%, indicating a relatively small deviation between numerical predictions and experimental measurements of water depth on the left side. Likewise, d_c shows an average error of 5.47%, indicating a moderate level of discrepancy between numerical model predictions and experimental data for the central water depth. On the other hand, d_r experiences an average error of 5.13%, reflecting a similar level of accuracy as d_c . These results confirm the influence of cube obstacles on water flow behavior and depth distribution. While the numerical model provides reasonable approximations, inconsistencies exist, likely due to complex interactions between water flow and obstructions. Observed deviations may stem from factors such as boundary conditions, geometric simplifications, or experimental limitations. The analysis highlights the importance of improving numerical models to better capture the complex dynamics of water flow around obstacles. By enhancing the fit between numerical predictions and experimental data, researchers can gain deeper insights into the behavior of water flow in obstacle-laden environments. This, in turn, contributes to more accurate forecasts and more informed decision-making in various fields, from hydraulic engineering to environmental management.

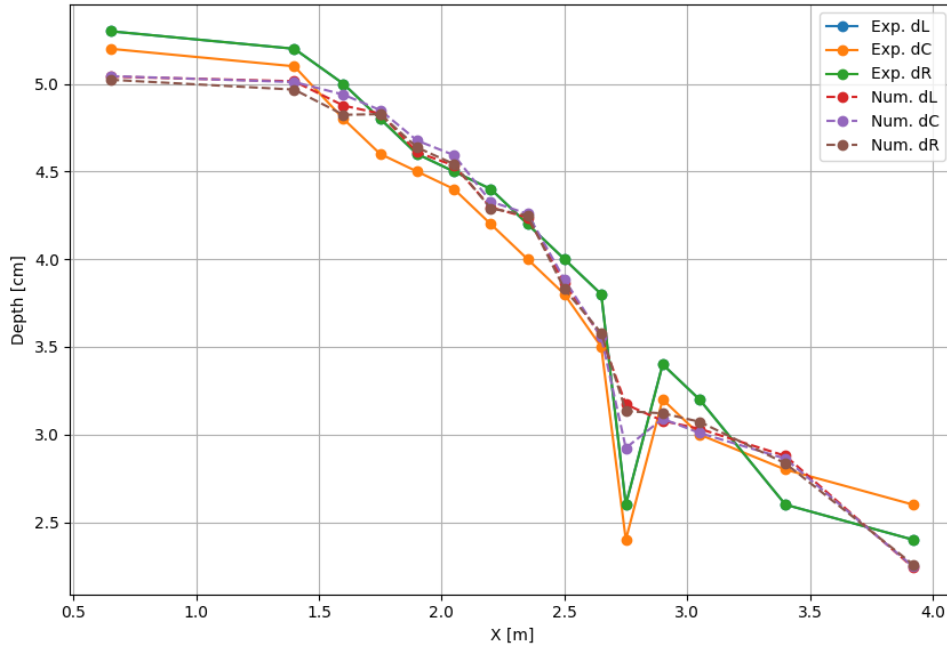


Figure 3.84. Water depth along channel for cubic obstacles; $\dot{Q} = 0.0035 \text{ m}^3/\text{s}$, $S = 0^\circ$

In Fig. 3.85, for $\dot{Q} = 0.004 \text{ m}^3/\text{s}$, $S = 0^\circ$, the results indicated a clear effect of cubic obstacles on the behavior and values of water runoff depth. Over the distance range studied, experimental values consistently show higher water flow depths compared to numerical simulations. This discrepancy indicates that the presence of cubic obstacles alters flow patterns, leading to increased water depths. Analysis of the average error rates illustrates the extent of these discrepancies. Average error rate for d_l to is about 11.97%, which indicates that the experimental values for the left side of the flow are, on average, about 12% higher than the numerical simulations. Likewise, the average error rate for d_c About 6.16%, which indicates a moderate difference between the experimental and numerical values of the flow center. Finally, the average error rate for d_r approximately 11.69%, which highlights a trend similar to d_l to with a little less contradiction. Understanding these error ratios is crucial to interpreting the reliability of numerical simulations compared to experimental observations. A higher mean error ratio indicates a greater deviation between the two data sets, indicating potential limitations or inaccuracies in the numerical model's representation of flow dynamics around cube obstacles.

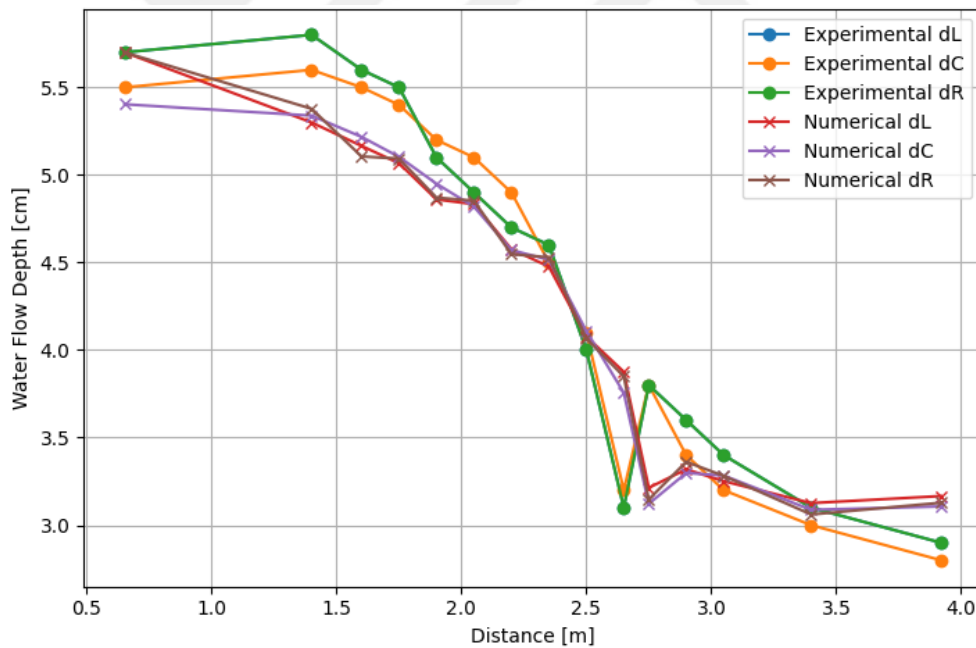


Figure 3.85. Water depth along channel for cubic obstacles; $\dot{Q} = 0.004 \text{ m}^3/\text{s}$, $S = 0^\circ$

The curves in Fig 3.86 for $\dot{Q} = 0.0045 \text{ m}^3/\text{s}$, $S = 0^\circ$ provide insight into how obstacles affect water depth and highlight any differences between experimental and numerical results. The presence of obstacles significantly affects the water depth, with experimental data generally showing slightly higher values compared to numerical simulations. Discrepancies between experimental and numerical values, especially near obstacle locations, suggest that factors such as

measurement accuracy and simplifications in the numerical model contribute to these differences. In all scenarios (d_l , d_c , and d_r), water depth decreases with increasing X , indicating a gradual decrease in water volume or flow rate downstream. While numerical simulations show smoother curves, experimental data exhibit fluctuations likely due to real-world complexities that are not fully captured in the simulations. The error ratios reveal the average deviation between experimental and numerical values for each depth component. Average error rate for d_l is about 11.97%, which indicates a large deviation between the experimental and numerical results. In comparison with, d_c . It shows a slightly lower deviation with an average percentage error of about 6.16%. Similarly, d_r . It shows an average error rate of about 11.69%, similar to d_l .

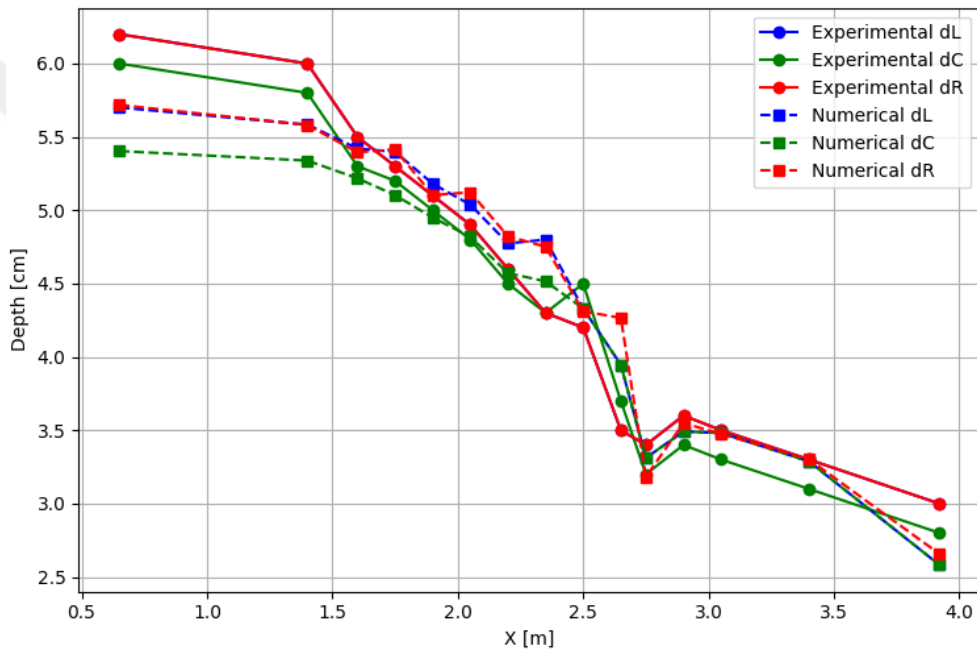


Figure 3.86. Water depth along channel for cubic obstacles; $\dot{Q} = 0.0045 \text{ m}^3/\text{s}$, $S = 0^\circ$

The results given in Fig. 3.87 for ; $\dot{Q} = 0.0035 \text{ m}^3/\text{s}$, $S = 1.0156^\circ$, confirm the significant influence of obstacles on water flow depth, which can be distinguished in both experimental and numerical data sets. As water moves downstream (X), a consistent pattern emerges: the depth of flow, characterized by d_l , d_c , and d_r steadily decreases. This phenomenon is a direct result of obstacles that impede the flow, cause turbulence, and change the course of the water. After examining the error percentage values, it turns out that the numerical model deviates from the experimental data with an average error percentage ranging between 13% to 14% across all depth components (d_l , d_c , and d_r).

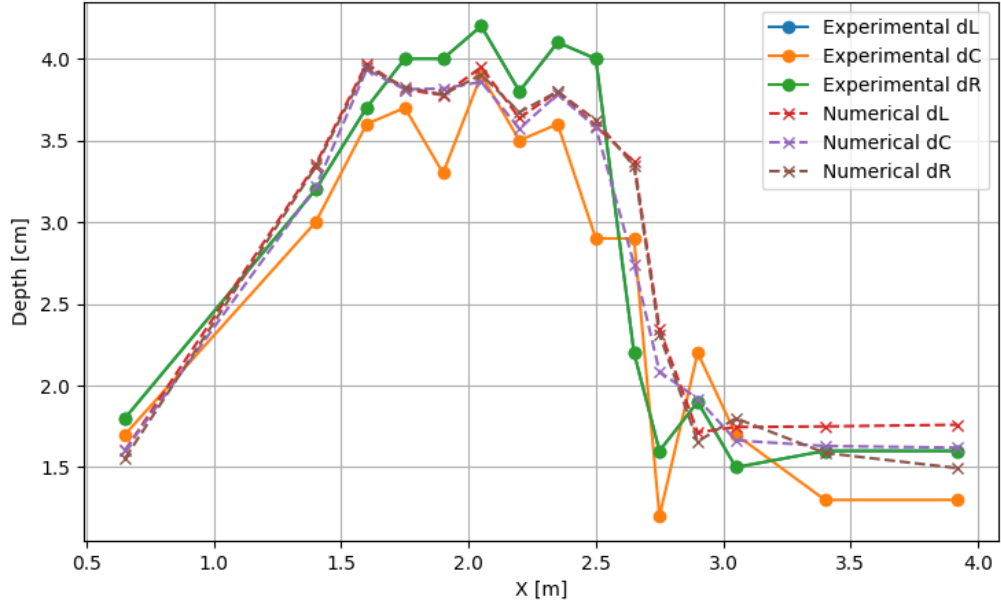


Figure 3.87. Water depth along channel for cubic obstacles; $\dot{Q} = 0.0035 \text{ m}^3/\text{s}$, $S = 1.0156^\circ$

The results in Fig. 3.88 for $\dot{Q} = 0.004 \text{ m}^3/\text{s}$, $S = 1.0156^\circ$ demonstrate the effect of obstructions on water flow behavior and depth, highlighting phenomena such as separation, eddy formation, and overall flow characteristics. When the flow encounters obstacles along its path, there are several observable effects. First, the presence of obstacles causes flow separation, where the flow separates from the surface of the obstacle. This phenomenon leads to the formation of recirculation zones or eddies behind obstacles, as evidenced by differences in water depth measurements (d_l , d_c , and d_r). Experimental and numerical data indicate a decrease in water depth below obstructions, highlighting the destructive effect of obstructions on flow continuity. Furthermore, numerical simulations show inconsistencies when compared to experimental observations, as evidenced by the percentage error values. The average percentage errors for each depth component (d_l , d_c , and d_r) range from about 16% to 21%. These discrepancies can arise from various factors, including simplifications or assumptions made in the numerical model, inaccuracies in experimental measurements, or differences in boundary conditions between simulated and realistic scenarios.

Overall, the results confirm the complex interplay between obstacles and water flow dynamics. Understanding these dynamics is critical for various applications, including hydraulic engineering, environmental modeling, and river management. Improving both experimental techniques and numerical models is essential to improve the accuracy of predictions and better understand the complex behavior of flows around obstacles.

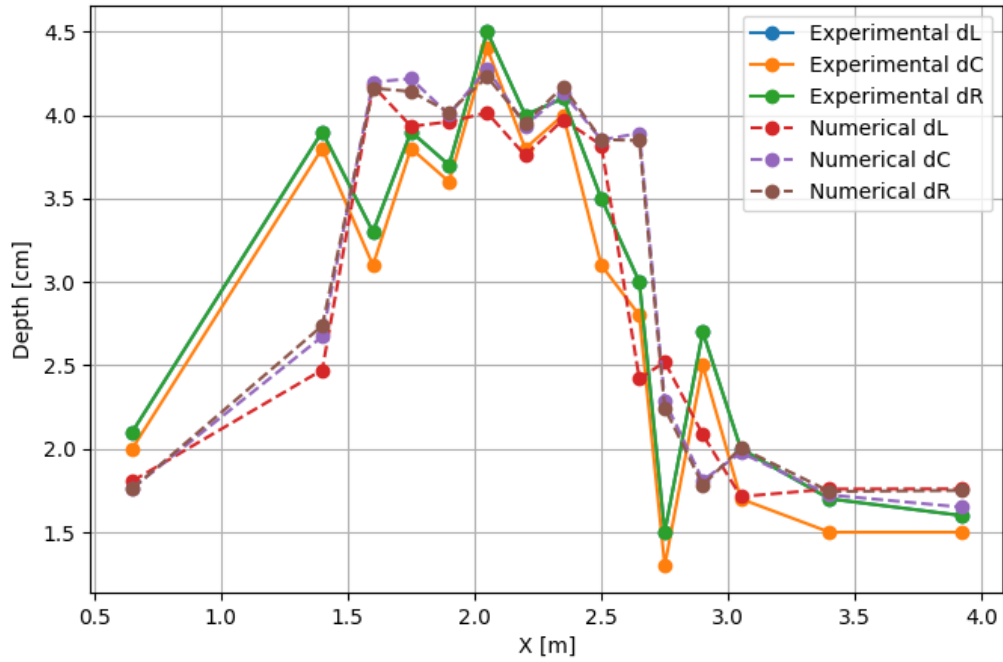


Figure 3.88. Water depth along channel for cubic obstacles; $\dot{Q} = 0.004 \text{ m}^3/\text{s}$, $S = 1.0156^\circ$

In examining the trends in Fig. 3.89, a consistent decrease in water depth with increasing distance for all three components was observed and this indicating the impact of obstacles on flow behavior. While the numerical simulation generally follows the experimental trend, deviations are noticeable, particularly in certain regions. These disparities could stem from inaccuracies in the numerical model, simplifications in the simulation, or uncertainties in experimental measurements. The average percentage error values, ranging from approximately 15% to 21%, highlight the discrepancy between the numerical and experimental datasets. These findings underscore the importance of refining numerical models and improving experimental techniques for better alignment between simulation and observation. Overall, the results emphasize the significant influence of obstacles on water flow depth and the need for continued validation and refinement of numerical models through experimental validation.

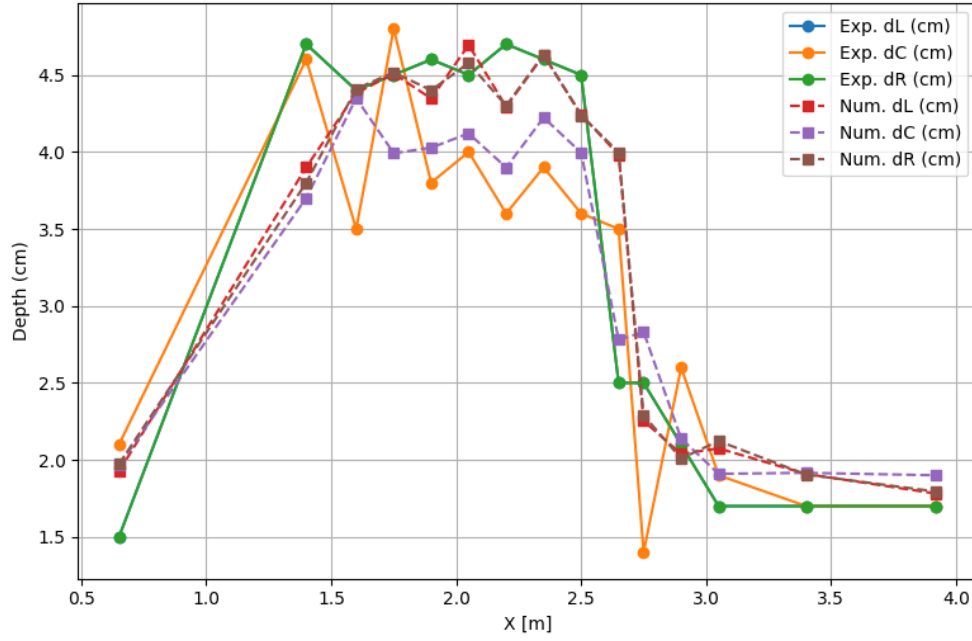


Figure 3.89. Water depth along channel for cubic obstacles; $\dot{Q} = 0.0045 \text{ m}^3/\text{s}$, $S = 1.0156^\circ$

The data in Fig. 3.90 for the case of $\dot{Q} = 0.0035 \text{ m}^3/\text{s}$, $S = 1.406^\circ$ show the effect of any obstruction on different flow characteristics downstream. When the flow encounters an obstacle, noticeable changes in flow depth, separation points and turbulence patterns appear, shaping the overall flow dynamics. The flow depths show a decreasing trend with increasing distance downstream. This decrease is a direct result of the obstruction created by the obstruction, which results in decreased flow depths. Numerical simulations generally agree with experimental observations, albeit with notable variations, as shown by the error ratios. Separation points (d_c) define where the flow begins to separate from the obstacle surface. These points hold importance in understanding flow behavior and are essential for designing structures to mitigate the effects of separation. At the bottom of the obstacle, the formation of vortices and turbulent regions becomes evident, affected by flow separation and obstruction.

The drag force exerted by the obstacle on the flow, together with the shear stress distribution, emphasizes the dynamic interaction between the flow and the obstacle surface. These factors are pivotal in engineering contexts, influencing structural design and hydraulic performance. Error ratio analysis is a vital tool in evaluating the accuracy of numerical models. Discrepancies between numerical predictions and experimental data, reflected in error ratios, identify areas that require model improvement or experimental verification. Identifying areas with notable deviations helps enhance model accuracy and predictive capabilities. In essence, the data provides valuable insights into the complex interplay between obstacles and flow dynamics.

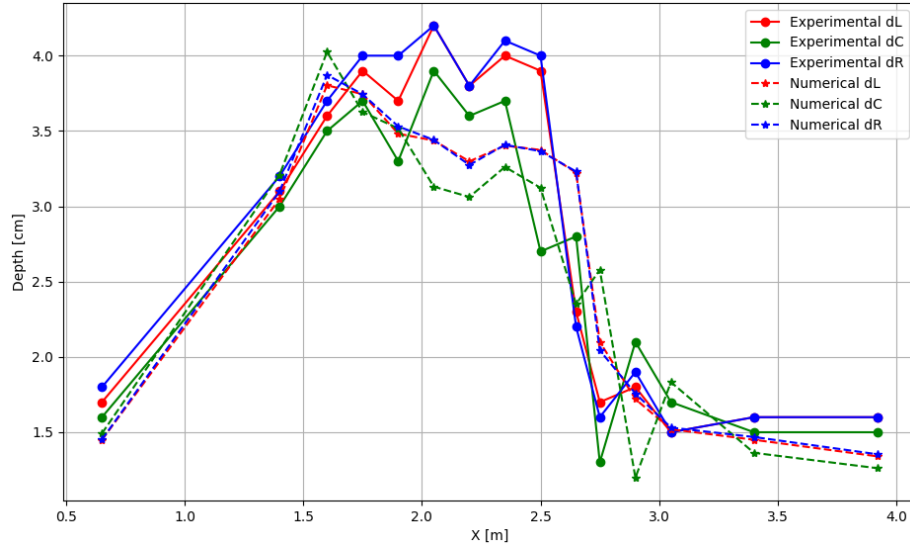


Figure 3.90. Water depth along channel for cubic obstacles; $\dot{Q} = 0.0035 \text{ m}^3/\text{s}$, $S = 1.406^\circ$

Through careful analysis of flow characteristics and error ratios, a deeper understanding of hydraulic phenomena emerges, guiding advances in numerical modeling and experimental techniques. The percentage error values provide quantitative insights into the agreement between experimental and numerical results for different flow characteristics. For the parameter d_l , which represents the flow depth, the average percentage error is calculated at 12.57%. This indicates that numerical predictions of flow depth, on average, deviate from experimental measurements by about 12.57%. Likewise, for d_c , which refers to the separation point where the flow separates from the obstacle surface, the average percentage error is higher at 18.96%. This indicates a relatively larger discrepancy between numerical simulations and experimental observations for this critical parameter. Finally, for d_r , which may correspond to the flow characteristics downstream of the obstacle, the average percentage error is 14.67%. Although it is somewhat lower than the error for d_c , it still highlights notable differences between numerical and experimental data for this aspect of flow behavior. These percentage error values serve as valuable metrics for evaluating the accuracy and reliability of numerical models. Understanding the magnitude of discrepancies helps identify areas for model improvement or experimental validation, ultimately enhancing our understanding of flow dynamics affected by obstacles.

The presented dataset for $\dot{Q} = 0.004 \text{ m}^3/\text{s}$, $S = 1.406^\circ$ in Fig. 3.91 provides insights into how obstruction affects various flow properties, such as flow depth, separation, vortices, drag force, and shear stress. Both experimental and numerical data depict changes in flow behavior downstream of the obstruction. The flow depth decreases when the flow encounters the obstacle, indicating the effect of its obstacle on the flow dynamics. However, there are notable differences

between experimental measurements and numerical predictions, as evidenced by the relatively high average percentage error for flow depth (15.76%).

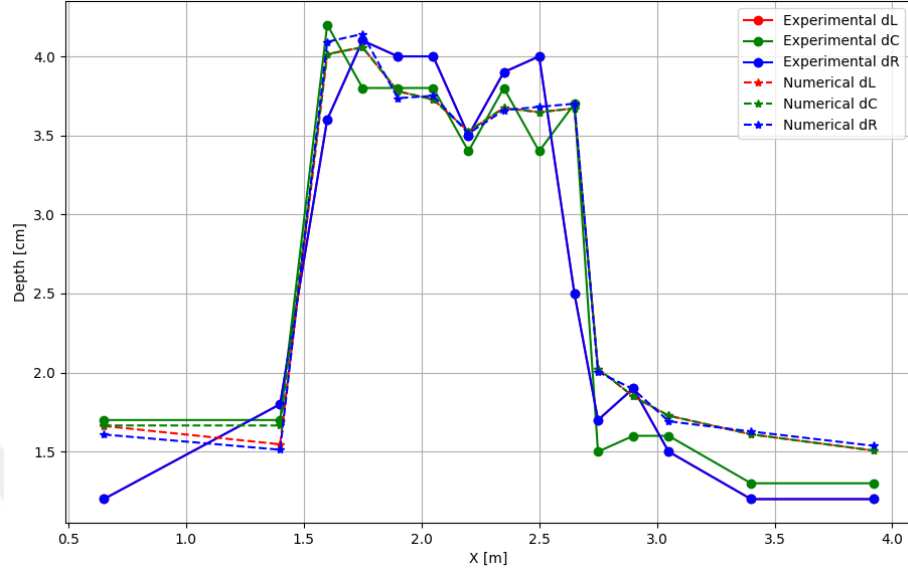


Figure 3.91. Water depth along channel for cubic obstacles; $\dot{Q} = 0.004 \text{ m}^3/\text{s}$, $S = 1.406^\circ$

Flow separation, characterized by changes in separation points, refers to the separation of the flow from the surface of the obstacle. The data reveal differences in cut-off points downstream, reflecting the complexity of flow dynamics affected by the obstruction. Vortices and turbulent regions form below the obstacle, contributing to increased mixing and energy dissipation. The drag force exerted by the obstacle on the flow changes the flow velocity profiles, while the shear stress distribution varies spatially and is affected by flow separation, vortices and turbulence. Error ratios provide quantitative measures of agreement between experimental and numerical results, highlighting significant discrepancies that call for further improvement of the numerical model or experimental procedures. Overall, the dataset emphasizes the complex interplay between obstacles and flow dynamics and directs I improvements in numerical modeling and experimental techniques.

The presented dataset in Fig. 3.92 for ; $\dot{Q} = 0.0045 \text{ m}^3/\text{s}$, $S = 1.406^\circ$ provides insights into how obstruction affects various flow properties, including flow depth, separation, vortex formation, drag force, and shear stress.

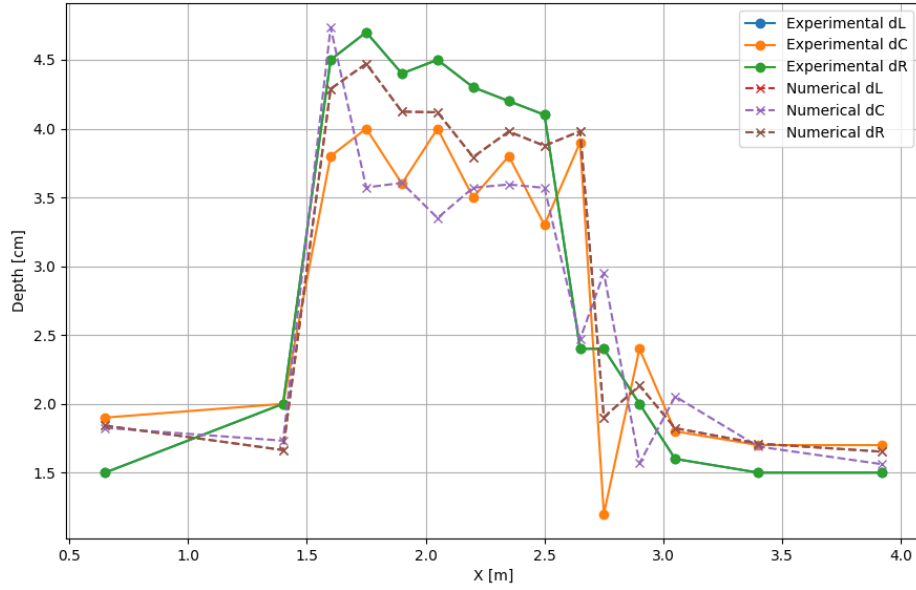
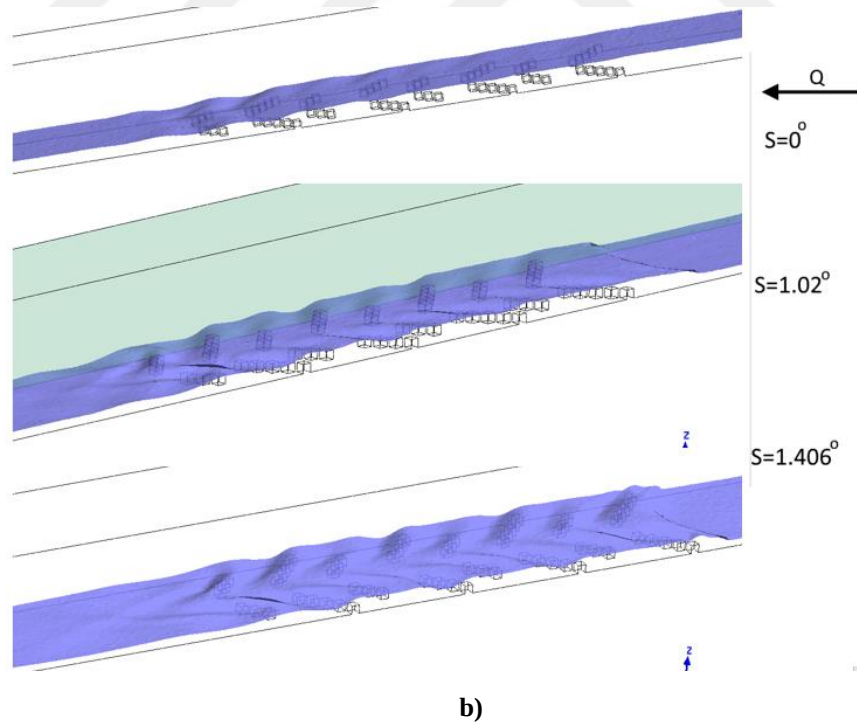
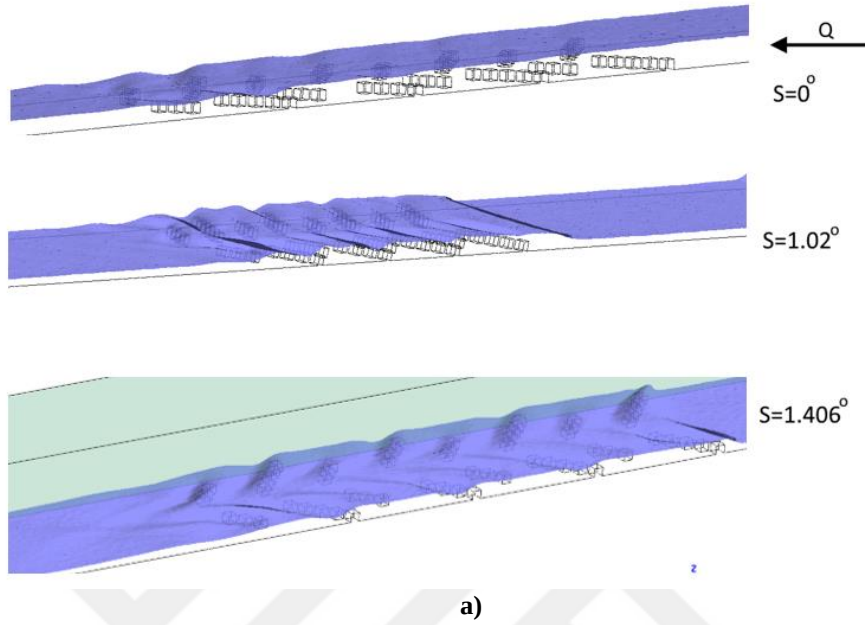


Figure 3.92. Water depth along channel for cubic obstacles; $\dot{Q} = 0.0045 \text{ m}^3/\text{s}$, $S = 1.406^\circ$

Experimental data indicate changes in flow depth when the flow encounters the obstacle. In general, the flow depth below the obstacle decreases, indicating the effect of the obstacle on the flow dynamics. However, there are differences between the experimental and numerical results, as evidenced by the high average percentage error for flow depth (14.52%). Flow separation occurs when the flow separates from the surface of the obstacle. The data show differences in separation points below the obstacle, reflecting the complex interaction between flow and obstacle geometry. Numerical simulations may provide insight into the precise location and extent of separation zones. Vortices and turbulent regions form beneath the obstacle, contributing to increased mixing and energy dissipation. These turbulent structures are evident in the flow properties observed in both experimental and numerical data.

The presence of the obstacle changes the flow velocity profile, generating drag force. This force acts on the obstacle and affects its motion and the flow field surrounding it. Experimental and numerical data provide valuable information about the magnitude and distribution of drag force, which helps in the design and optimization of structures in liquid environments. Shear stress, the force per unit area exerted by moving fluid molecules, varies spatially and is affected by flow separation, vortices, and turbulence. Both experimental and numerical data provide insights into the shear stress distribution around the obstacle, helping to evaluate its impact on the surrounding environment and structures. The average error calculated for each flow characteristic (d_l , d_c , d_r) highlights the agreement between experimental and numerical results. The relatively high average percentage error of d_c (21.64%) indicates significant discrepancies between the two data sets, indicating potential areas for improvement in numerical modeling or experimental procedures.

Some numerical results of depth analysis are given by Figs.3.93 (a-c) representing the depths for the cube obstacles.



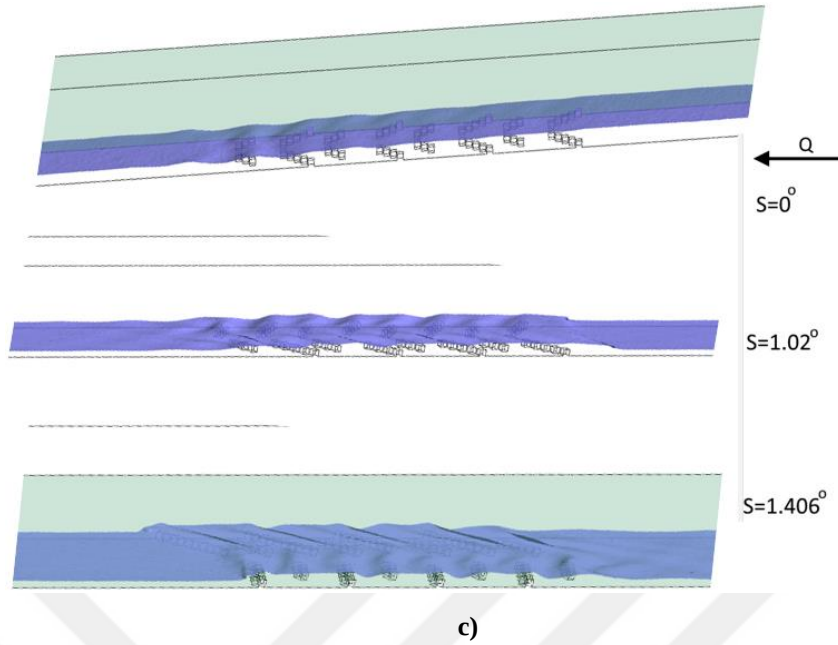


Figure 3.93. Depth for the rectangular arc obstacle, from the side view for; **a)** $\dot{Q} = 0.0035 \text{ m}^3/\text{s}$; **b)** $\dot{Q} = 0.004 \text{ m}^3/\text{s}$; **c)** $\dot{Q} = 0.0045 \text{ m}^3/\text{s}$

3.3.4. Depth Analysis along Channel Distance with Elliptic Obstacles (Experimental Only)

Analysis of data for $0.0035 \text{ m}^3/\text{s}$ discharge sheds light on the behavior of depth with elliptical obstacles across different slope conditions. As it is seen from Fig. 3.94, across different stations, fluctuations in both flow velocities were observed (N.VL, N.VC, N.VR) and depths (N.dL, N.dC, N.dR). In general, depths tend to be deeper near the center of the channel (N.dC) than at the edges (N.dL and N.dR). This trend is affected by the convergence of the flow towards the center, which leads to higher depths. However, the presence of elliptical obstacles introduces complications, affecting flow patterns and changing depth distributions. Areas where obstructions impede flow, such as areas near banks or around obstructions themselves, may experience reduced depths due to increased turbulence or flow acceleration. Conversely, at the bottom of obstructions or in areas where flow converges, depths may increase as water accumulates or slows. The influence of slope conditions on depth behavior is notable. Gentle slopes promote slower flow speeds, allowing water to accumulate and deepen the channel. In contrast, steeper slopes may result in faster water flow, resulting in generally shallower depths. These observations underscore the complex relationship between channel geometry, slope conditions, and the presence of obstacles. These insights are essential for hydraulic analysis in river engineering and management, underscoring the need for comprehensive assessments to accurately understand and predict flow behavior.

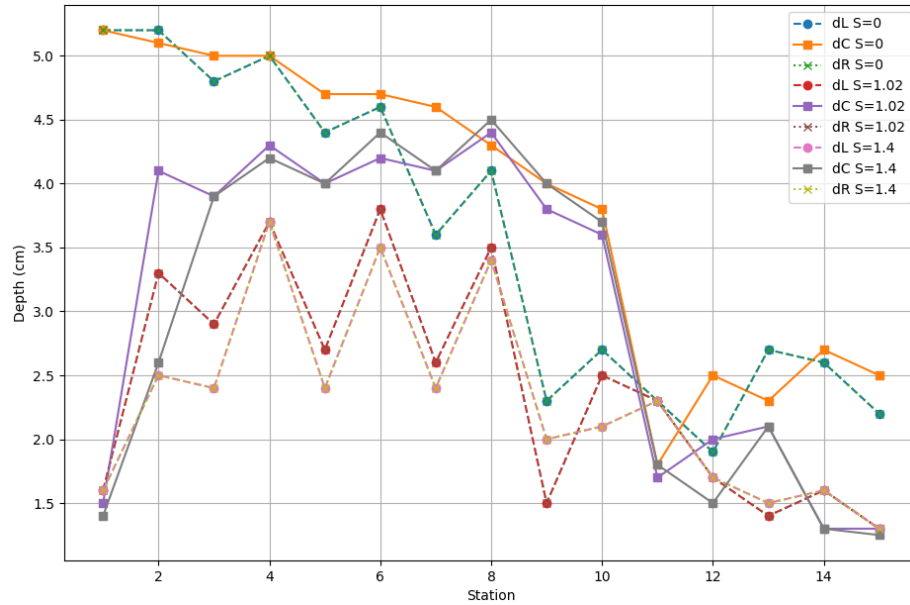


Figure 3.94. Variation of water flow depth for elliptical obstacle $\dot{Q} = 0.0035 \text{ m}^3/\text{s}$

Figure 3.95 shows the data for $0.004 \text{ m}^3/\text{s}$ discharge reveals interesting insights into depth behavior with elliptical obstacles across different slope conditions. Across different stations, study observe fluctuations in both flow velocities (N.VL, N.VC, N.VR) and depths (N.dL, N.dC, N.dR). In general, deeper water tends to be located closer to the center of the channel (N.dC) than to the edges (N.dL and N.dR). This can be attributed to the convergence of the flow towards the center, leading to higher depths. However, the presence of elliptical obstacles introduces complications, affecting flow patterns and changing depth distributions. In areas where obstructions impede flow, such as areas near banks or around obstructions themselves, depths may decrease due to increased turbulence or acceleration of the flow. Conversely, at the bottom of obstructions or in areas where flow converges, depths may increase as water accumulates or slows. Furthermore, differences in slope conditions play an important role in depth behavior. For example, gentler slopes tend to promote slower flow velocities, allowing water to accumulate and deepen the channel. In contrast, steeper slopes may cause water to flow faster, resulting in generally shallower depths. These observations underscore the complex interplay between channel geometry, slope conditions, and the presence of obstacles. These insights are essential for hydraulic analysis in river engineering and management, highlighting the need for comprehensive assessments to accurately understand and predict flow behavior.

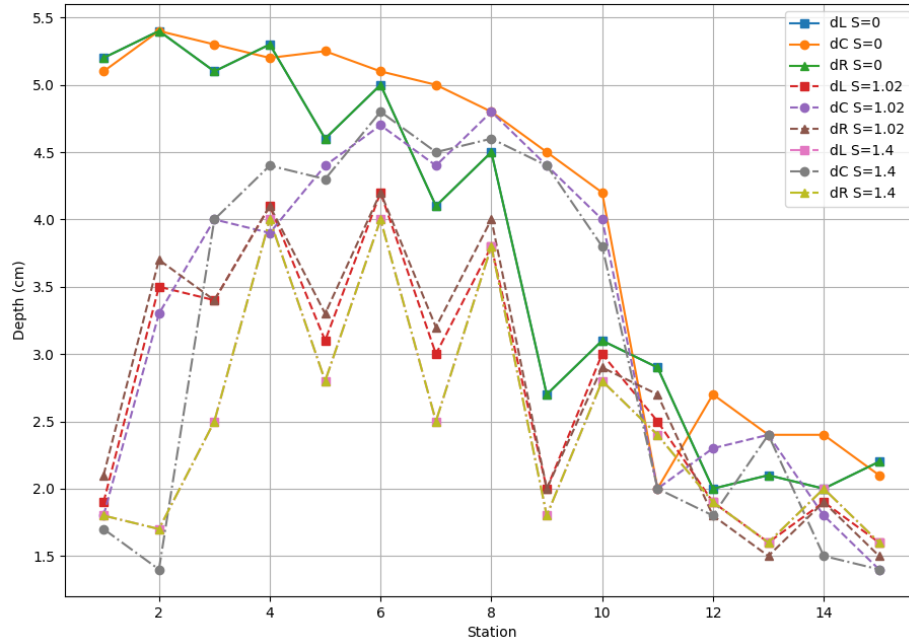


Figure 3.95. Variation of water flow depth for elliptical obstacle $\dot{Q} = 0.004 \text{ m}^3/\text{s}$

The data presented in Fig. 3.96 provide valuable insights into how elliptical obstacles affect depth behavior across different slope conditions of 0° , 1.02° , and 1.406° . By examining depth measurements at different stations of these slopes, research can discern patterns and differences in flow dynamics. Across stations, changes in depth measurements has been observed (N.dL, N.dC, N.dR) in response to the presence of elliptical obstacles. In general, depths tend to vary, with deeper water more often observed near the center of the channel (N.dC) than at the edges (N.dL and N.dR). This phenomenon can be attributed to the convergence of the flow towards the center, causing higher depths. Furthermore, differences in slope conditions introduce additional complications to depth behavior. For example, steeper slopes (1.406°) may cause water to flow faster, resulting in generally shallower depths. Conversely, gentler slopes (0°) may promote slower flow velocities, allowing water to accumulate, thus deepening the channel. The presence of elliptical obstacles interacts with slope conditions, altering flow patterns and thus affecting depth distribution. In areas where obstructions impede flow, such as areas near banks or around obstructions themselves, depths may decrease due to increased turbulence or acceleration of the flow. Conversely, at the bottom of obstructions or in areas where flow converges, depths may increase as water accumulates or slows. These depth variations emphasize the complex interplay between channel geometry, slope conditions, and the presence of obstacles, highlighting the need for comprehensive hydraulic analysis in river engineering and management contexts.

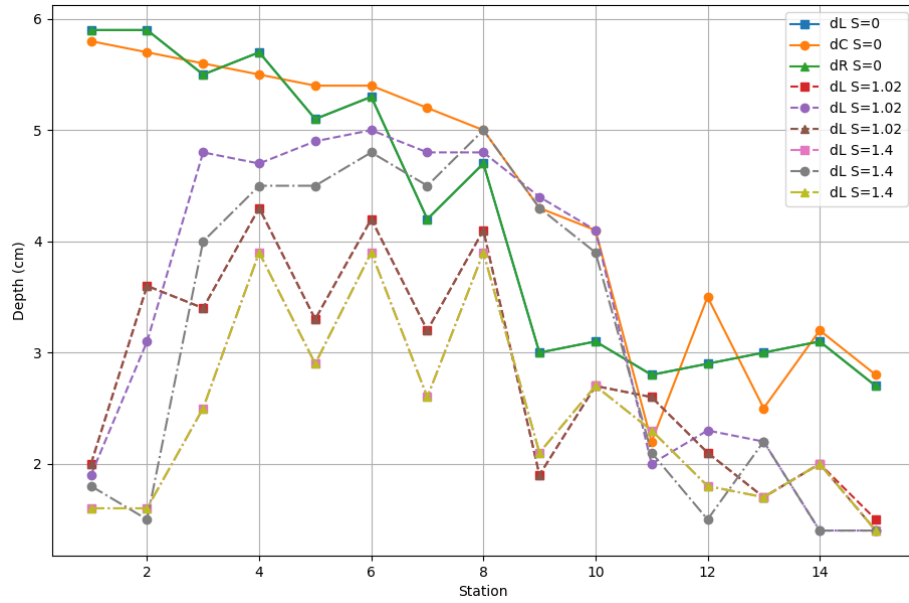


Figure 3.96. Variation of water flow depth for elliptical obstacle $\dot{Q} = 0.004 \text{ m}^3/\text{s}$

3.3.5. Depth Analysis along Channel Distance with Cylindrical Obstacles (Experimental Only)

In Fig. 3.99 the water velocity and depth of the water are given in same graphs for all tested flowrates and slopes.

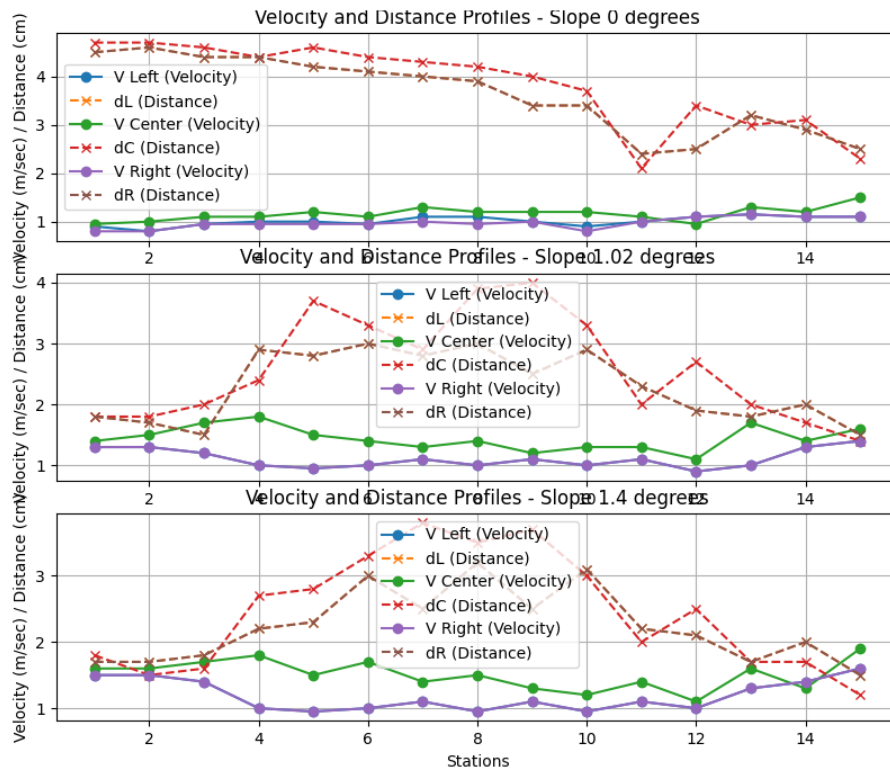


Figure 3.97. Velocity and depth values for cylindrical obstacle

3.4. Froude (Fr) Numbers of Newtonian Fluid Flow

3.4.1. Importance of Froude Numbers in Open Channel Flows

In the world of open-channel hydraulics, understanding the behavior of flowing water is critical for various engineering applications, from flood management to riparian habitat restoration. Among the large group of parameters used to describe open channel flow, the Froude number stands out as a fundamental dimensionless quantity that has significant implications for flow evaluation, especially in the presence of obstacles. The Froude number, denoted by Fr , is a key metric in evaluating the relative importance of inertial forces to gravitational forces within flowing water. The Froude number is defined as the ratio of flow velocity to the square root of the product of the gravitational acceleration and the characteristic length scale and provides important insights into flow dynamics and stability. In open-channel hydraulics, where flow occurs in natural or man-made channels with free surfaces exposed to atmospheric pressure, the Froude number emerges as a crucial tool for engineers and researchers alike. One of the most compelling scenarios in open channel flow analysis involves obstructions within the channel. These obstacles can appear in various forms, ranging from natural features such as rocks and plants to engineered structures such as bridge piers and hydraulic structures. Understanding the interaction between flowing water and these obstacles is essential to predict flow patterns, identify potential hazards, and design hydraulic structures that are resistant to various hydraulic conditions. The importance of the Froude number in the presence of obstacles cannot be overstated. It plays a pivotal role in determining the transition between different flow regimes, such as subcritical, critical and supercritical flow, which have distinct hydraulic properties and behaviors. By evaluating the Froude number, engineers can evaluate hydraulic jumps, flow separation, and energy dissipation that occur around obstructions, which helps in designing efficient and safe hydraulic structures. Furthermore, the Froude number facilitates the prediction of flow velocities, depths, and pressures around obstacles, thus guiding decision-making processes related to flood risk management, sediment transport, and ecological restoration in riverine environments. Its utility extends to various fields, including civil engineering, environmental sciences, and hydraulic modelling, underscoring its interdisciplinary importance in addressing complex hydraulic challenges. In this context, this paper aims to explore the importance of the Froude number in evaluating flow in open channels, with a particular focus on its application in scenarios involving obstructions. Through comprehensive analysis of experimental and numerical data, aimed to clarify the role of the Froude number in characterizing flow behavior and guiding engineering practices in open channel hydraulics. In addition, it is necessary to understand how the Froude number (Fr) is calculated.

$$Fr = \frac{V}{\sqrt{gy}} \quad (3.1)$$

where: V is the flow velocity, g is the gravitational acceleration, d is the flow depth. By specifying the flow velocity and the characteristic depth or length scale, engineers can easily calculate the Froude number, providing valuable insights into flow dynamics and behavior in open channel hydraulics.

3.4.2. Fr numbers with Rectangle Obstacle

Analysis of experimental and numerical results of the Fr number along an open channel with rectangular obstacles at a slope of 0° and varying flow rates Q of $0.0035 \text{ m}^3/\text{s}$, $0.04 \text{ m}^3/\text{s}$ and $0.045 \text{ m}^3/\text{s}$ provides insight into the flow. Properties at different points downstream. The comparison is made for the left, center, and right positions across the channel at specified distances from the source boundaries.

For a flow rate of $0.0035 \text{ m}^3/\text{s}$, the Fr number values are initially low, indicating subcritical flow conditions. At the upstream location of 0.6509 m , the Fr values are around 0.30 to 0.34 , with slight differences between the experimental and numerical results. As the location move to 1.4 m , there is a slight increase in the numerical results, indicating a slightly accelerated flow compared to experimental observations. From 1.6 m to 2.05 m , the experimental and numerical Fr values increase, especially in the center, indicating a transition towards critical flow due to the impact of obstacles.

At 2.2 m and 2.35 m , there are peaks in the Fr values, with the experimental data showing a maximum of about 0.67 at the center at 2.5 m . Numerical results follow a similar pattern but tend to be slightly higher. This peak indicates significant flow disturbance and acceleration due to obstructions. After this point, at 2.65 m to 2.9 m , the highest Fr values are observed, indicating near-supercritical conditions. This section highlights the critical flow dynamics and energy changes in the channel.

For a flow rate of $0.004 \text{ m}^3/\text{s}$, the Fr number remains initially low at 0.6509 m , with values ranging from 0.30 to 0.34 . However, as the distance increases, the Fr number values rise significantly, especially in the numerical results, indicating a more pronounced effect of obstacles on the flow. At 1.4 m , the digital Fr values are about 1.54 , compared to experimental values of about 0.34 . This discrepancy may be due to ideal conditions in the numerical simulations.

At 1.75 m , the experimental Fr values reach around 0.43 , while the numerical values are much higher, around 1.75 . This significant variation continues along the channel, with peaks observed in the numerical results at different points. For example, at 2.05 m , the experimental center value is about 0.47 , while the numerical value is about 1.94 , highlighting the critical

influence of obstacles in numerical simulations. The trend of higher Fr numerical values continues downstream, indicating potentially supercritical conditions influenced by obstacles.

For a flow rate of $0.0045 \text{ m}^3/\text{s}$, the Fr number remains initially low at 0.6509 m , around 0.32 for both the experimental and numerical results. However, as the flow moves downstream, the numerical values increase significantly, indicating a strong acceleration of the flow. At 1.4 m , digital Fr values are around 0.42 , compared to experimental values of around 0.38 . This trend continues, with numerical values generally being higher than experimental values.

At 1.75 m , the empirical center value is about 0.41 , while the numerical value is also about 0.41 , indicating some agreement between the two methods. However, at 2.2 m and beyond, the numerical results show much higher Fr values, indicating stronger effects of obstacles. For example, at a height of 2.65 meters , the numerical values are around 0.65 , compared to experimental values of around 0.67 . This pattern continues, with digital Fr values peaking around 0.72 at 2.75 m and declining slightly after this point.

In summary, analysis of the Fr number along a channel with rectangular obstacles and varying flow rates shows significant differences between experimental and numerical results. While experimental data provide practical insight into flow conditions, numerical simulations often predict higher Froude numbers, indicating stronger flow acceleration and turbulence due to obstacles. These differences highlight the importance of considering both experimental and numerical approaches in understanding flow dynamics and designing hydraulic structures.

For $\dot{Q} = 0.0035 \text{ m}^3/\text{s}$, $S = 1.0156^\circ$ at 0.6509 m , the empirical Fr numbers on the left and right (0.894) are lower than the center (1.195), indicating an acceleration of the central flow. The numerical results show a similar pattern, with slightly higher values for the center (1.247). At 1.4 m , experimental and numerical results show a decrease, indicating a decrease in flow velocity, with values of around 0.51 for the experimental left and right, and 0.60 for the center. The numerical values are slightly lower (0.54).

At 1.6 m , the value of the experimental center (0.665) is much higher than the left (0.422) and right (0.369), indicating channel flow disturbances. The numerical values are similar, indicating consistent model accuracy. At 1.75 m , the experimental Fr values increase, especially at the center (0.706). Numerical values correspond to this increment. At 1.9 m , the experimental center value peaks at 0.721 , while the numerical values show slightly lower results (0.743), indicating the region of maximum flow velocity affected by the obstruction. From 2.05 m to 2.2 m , the experimental and numerical results were consistent, with the experimental values being around 0.657 to 0.730 and the numerical values slightly lower. At 2.35 m , the experimental values decrease significantly, indicating lower flow velocities, while the numerical results indicate slight differences. From 2.5 m to 2.75 m , the Fr values increase, indicating acceleration

due to channel slope and obstructions. The numerical results match closely, especially at 2.75 m, where the beta center value is 1.045 and the digital is slightly higher (1.058).

At 2.9 m, higher experimental Fr values (1.101) compared to numerical (1.199) indicate the critical flow region. At 3.05 m, the experimental Fr values continue to rise (1.276), while the numerical values show a similar trend (1.405). From 3.4 m to 3.92 m, the experimental Fr peaks reach 1.451 at 3.92 m, and the numerical results show slightly higher peaks (1.516).

Summary of 0.0035 m³/s, the flow is initially below the critical level and moves to near critical obstructions. The numerical results generally follow experimental trends, with slight overestimation. Center values are consistently higher due to channel steepness and obstructions, indicating flow acceleration and turbulence.

When the flow rate is 0.004 m³/s, at the initial section (0.6509 m to 1.4 m), especially at 0.6509 m, the experimental values are higher at the center (1.316), indicating a significant acceleration. The numerical values are slightly higher (1.228). At 1.4 m, both data sets show decreasing Fr values, indicating stabilization, with numerical results (0.658) closely following experimental trends.

At center section (1.6 m to 2.75 m), from 1.6 m to 1.75 m, the experimental and numerical Fr values peak at the center due to obstacles. The experimental values range between 0.749 and 0.665, with the numerical results being consistent. From 2.05 m to 2.35 m, the experimental values indicate slight fluctuations, while the numerical values indicate similar trends. At 2.5 m, the experimental Fr values are high (0.739) with numerical values showing slightly lower trends. At 2.75 m, peak values are observed, with the experimental at 1.100 and the numerical alignment closely aligned.

At lower section (2.9 m to 3.92 m), from 2.9 m to 3.4 m, both data sets show rising Fr values, indicating increasing flow velocities. At 3.92 m, the maximum experimental Fr value is 1.451, with slightly lower numerical results (1.516).

If, 0.004 m³/s case summarized, it will be found that the flow shows similar trends as 0.0035 m³/s with higher peaks due to increased discharge. The numerical results match the experimental data closely, indicating reliable simulation accuracy.

Finally, when the flow rate is 0.0045 m³/s, at initial section (0.6509 m to 1.4 m), especially at 0.6509 m, higher experimental center values (0.725) indicate acceleration, with numerical results showing similar trends. At 1.4 m, both data sets show decreasing Fr values, indicating stability with slight numerical overestimation.

When the center section (1.6 m to 2.75 m) handled, it seemed that from 1.6 m to 1.75 m, the peak values at the center are due to obstructions, with experimental values ranging from 0.704 to 0.642, and the numerical values are closely related. At 2.05 m, the experimental Fr values stabilize around 0.613, with the numerical results holding constant. From 2.35 m to 2.65 m, slight

fluctuations are observed, with experimental values ranging from about 0.704 to 0.639, and numerical values showing similar trends. At 2.75 m, maximum experimental values are observed at 1.046, with numerical values following closely behind.

At the lower section (2.9m to 3.92m), at about 2.9 m, high Fr values indicate critical flow regions, with numerical results closely matching the experimental data. From 3.05 m to 3.92 m, both data sets show increasing Fr values, indicating high flow velocities, with experimental values peaking at 1.363.

For $\dot{Q} = 0.0035 \text{ m}^3/\text{s}$, $S = 1.406^\circ$, at the initial section (0.6509m to 1.4m) At 0.6509 m, the experimental Fr numbers on the left and right (0.896) are lower than the center (1.033), indicating an acceleration of the central flow. The numerical results follow a similar pattern, with the center values (1.042) being slightly higher than the experimental results. At 1.4 m, the experimental and numerical results show a decrease in Fr , indicating a decrease in flow velocity, with the experimental values being around 0.672 and the numerical values around 0.847.

At the center section (1.6m to 2.75m), at 1.6 m, the value of the experimental center (0.584) is higher than the left (0.393) and right (0.393), indicating flow disturbances. The numerical values are consistent, showing slightly lower results (0.554 for the center). At a height of 1.75 m, the experimental Fr values increase, especially at the center (0.653). The numerical results match this increase closely. By 1.9 m, the experimental center value reaches 0.718, while the numerical results show a lower value (0.695), indicating the region of maximum flow velocity affected by the obstruction. From 2.05 m to 2.2 m, the experimental and numerical results were consistent, with the experimental values being around 0.688 to 0.749 and the numerical values slightly lower. At 2.35 m, the experimental values decrease, indicating lower flow velocities, while the numerical results indicate slight differences. From 2.5 m to 2.75 m, the Fr values increase, indicating acceleration due to channel slope and obstructions. The numerical results match closely, especially at 2.75 m, where the beta center value is 1.066 and the digital is slightly higher (1.054).

At the lower section (2.9m to 3.92m), at 2.9 m, higher experimental Fr values (1.059) compared to numerical (1.156) indicate a critical flow region. At 3.05 m, the experimental Fr values continue to rise (1.181), while the numerical values show a similar trend (1.500). From 3.4 m to 3.92 m, the experimental Fr peak reaches 1.257 at 3.92 m, and the numerical results show slightly higher peaks (1.506).

Summary for $0.0035 \text{ m}^3/\text{s}$; the flow is initially below the critical level, and moves to near critical obstructions. The numerical results generally follow experimental trends, with slight overestimation. Center values are consistently higher due to channel steepness and obstructions, indicating flow acceleration and turbulence.

For $\dot{Q} = 0.004 \text{ m}^3/\text{s}$, $S = 1.406^\circ$ at initial section, i.e. at 0.6509 m, the experimental values are higher at the center (1.271), indicating a significant acceleration. The numerical values are higher (1.783). At 1.4 m, both data sets show decreasing Fr values, indicating stabilization, with numerical results (1.182) closely following experimental trends.

At the center section (1.6m to 2.75m), from 1.6 m to 1.75 m, the experimental and numerical Fr values peak at the center due to obstacles. The experimental values range between 0.769 and 0.681, with the numerical results holding constant. From 2.05 m to 2.35 m, the experimental values indicate slight fluctuations, while the numerical values indicate similar trends. At 2.5 m, the high experimental Fr values (0.729) with numerical values show slightly higher trends. At 2.75 m, peak values are observed, with the experimental at 1.280 and the numerical alignment closely aligned.

At lower section (2.9 m to 3.92 m), from 2.9 m to 3.4 m, both data sets show rising Fr values, indicating increasing flow velocities. At 3.92 m, the maximum experimental Fr value is 1.234, with slightly higher numerical results (1.390).

Summary for $0.004 \text{ m}^3/\text{s}$; the flow shows similar trends as $0.0035 \text{ m}^3/\text{s}$, with higher peaks due to increased discharge. The numerical results match the experimental data closely, indicating reliable simulation accuracy.

For the case of $\dot{Q} = 0.0045 \text{ m}^3/\text{s}$, $S = 1.406^\circ$, at the initial section (0.6509 m to 1.4 m), at 0.6509 m, higher experimental center values (1.480) indicate acceleration, with numerical results showing much higher values (7.874). At 1.4 m, both data sets show decreasing Fr values, indicating stability, with slight numerical overestimation.

For center section (1.6m to 2.75m), from 1.6 m to 1.75 m, the peak values in the center are due to obstructions, with experimental values ranging from about 0.923 to 0.774, and numerical values much higher (16.310). At 2.05 m, the experimental Fr values stabilize around 0.666, with the numerical results holding constant. From 2.35 m to 2.65 m, slight fluctuations were observed, where the experimental values ranged from about 0.830 to 0.643, and the numerical values showed much higher trends. At 2.75 m, maximum experimental values of 1.418 are observed, with much higher numerical values (24.646).

At the lower section (2.9 m to 3.92 m), at 2.9 m, high Fr values indicate critical flow regions, with numerical results closely matching the experimental data. From 3.05 m to 3.92 m, both data sets show an increase in Fr values, indicating high flow velocities, with experimental values peaking at 1.480 at 3.92 m

3.4.3. Fr Numbers with Rectangular Arc Obstacles

At the initial section, for the case of $\dot{Q} = 0.0035 \text{ m}^3/\text{s}$, $S = 0^\circ$, at 0.6509 m, the experimental Fr numbers are slightly higher in the center (0.363) compared to the sides (0.323), indicating higher flow velocities in the center. The numerical results are very close to the experimental results, with the central value being 0.325. At 1.4 m, the experimental and numerical Fr values are quite consistent, showing slightly higher centrality values, indicating stable flow behavior.

When center section (1.6 m to 2.75 m) is taken into account, it is seen that at 1.6 m, the experimental Fr values are higher at the center (0.408) than at the sides (0.369), indicating increased flow velocity at the center. The numerical results agree closely with the experimental results. At 1.75 m, the experimental values show a small drop in the center (0.406), while the numerical results remain stable. Moving to 1.9 m, the experimental values rise slightly at the center (0.454), while the numerical results are slightly lower but follow the same trend. At 2.05 m and 2.2 m, the experimental Fr values at the central peak are 0.464 and 0.507, respectively, indicating an increase in flow velocity. The numerical results agree well, with minor differences. From 2.35 m to 2.5 m, the experimental and numerical values continue to show consistent trends with slight fluctuations. At 2.65 m, the experimental Fr values at the center are lower (0.629) compared to the sides (0.631), indicating a more uniform flow distribution. The numerical results closely match this trend. At 2.75 m, the experimental values show a slight decrease in the center (0.618), while the numerical values are higher in the center (0.648), indicating a difference in flow dynamics around this point.

At the lower section (2.9 m to 3.92 m), at 2.9 m, the experimental Fr values at the center (0.582) are lower than at the sides, indicating a lower flow velocity. Numerical results show a similar trend. At 3.05 m, both data sets show higher Fr values, indicating increased flow velocities, with numerical results slightly lower than the experimental results. From 3.4 m to 3.92 m, the experimental Fr values at the center rise to a peak of 0.646, while the numerical results show a similar trend with a slight overestimation.

Summary for $0.0035 \text{ m}^3/\text{s}$, the experimental and numerical results closely match, with slight numerical overestimation. The flow shows higher velocities in the center, and transitions to a more regular flow downstream.

With increasing flowrate to $0.004 \text{ m}^3/\text{s}$ at initial section (0.6509 m to 1.4 m), at 0.6509 m, the experimental Fr values at the center (0.413) are slightly higher than at the sides, indicating an increased flow velocity at the center. The numerical results closely match, with minor deviations. By 1.4 m, the experimental and numerical values show a decrease, indicating stabilization of the flow.

For the center section (1.6 m to 2.75 m), at 1.6 m, the experimental Fr values at the center (0.435) are higher than at the sides, indicating increased flow velocity at the center. Numerical

results show similar trends. At 1.75 m, the experimental values decrease slightly at the center (0.423), while the numerical results remain stable. Moving to 1.9 m, the experimental Fr values at the center rise to 0.494, indicating higher flow velocities. The numerical results are slightly lower but follow the same trend. At 2.05 m and 2.2 m, the experimental Fr values at the central peak show an increase in flow velocities, with the numerical results closely matching. From 2.35 m to 2.5 m, the experimental and numerical values continue to show consistent trends with slight fluctuations. At 2.65 m, the experimental Fr values are higher at the center (0.649), while the numerical values closely match this trend. At 2.75 m, both data sets show a slight dip in the center, indicating lower flow velocities.

Lastly at the lower section (2.9 m to 3.92 m), at 2.9 m, the experimental Fr values at the center are higher, indicating increased flow velocities. Numerical results follow closely. From 3.05 m to 3.92 m, both experimental and numerical values show an increasing trend, indicating increasing flow velocities downstream, with the numerical values closely matching. Hence, for $0.004 \text{ m}^3/\text{s}$, the experimental and numerical results show a consistent trend, with slight numerical overestimation. The flow dynamics are similar to the low flow rate, with higher velocities at the center.

When $\dot{Q} = 0.0045 \text{ m}^3/\text{s}$, $S = 0^\circ$, at 0.6509 m, the experimental Fr values are consistent, with slightly higher values in the center. The numerical results match closely. At 1.4 m, the experimental and numerical values show stability, with consistent trends. Center section (1.6m to 2.75m) at 1.6 m, the experimental Fr values rise, indicating increased flow velocities. The numerical results are slightly lower but follow the same trend. When the slope is $S = 1.0156^\circ$ and the flow rate is $0.035 \text{ m}^3/\text{s}$, the experimental Fr numbers at the left, center, and right positions are 0.870, 0.985, and 0.870, respectively. The corresponding numerical Fr numbers are 0.939, 1.032, and 0.985. The data proceed similarly for different values of the Fr numbers. By comparing experimental and numerical results, accuracy can be evaluate the of the numerical model used to simulate flow behavior. Ideally, the numerical results should closely match the experimental data. Fr numbers vary at different positions (left, center, and right) along the obstacle. This indicates that the flow properties change spatially, perhaps due to the interaction of the flow with the obstacle.

As the discharge increases from $0.0035 \text{ m}^3/\text{s}$ to $0.0045 \text{ m}^3/\text{s}$ the Fr numbers generally tend to increase. This indicates higher flow velocities and potentially more turbulent flow conditions.

The study of the Fr in the context of a rectangular arc obstacle with an inclination of 1.406° and varying flow rates provides valuable insights into the flow dynamics around the obstacle. Fr values are generally higher in the center and right sections, indicating higher flow velocities or lower depths at these points. Numerical simulations also show a similar pattern, with the highest Fr numbers typically at the center. The numerical results provide a smoother gradient

of Fr values, indicating that flow modeling captures the general trend but may slightly underestimate or overestimate specific local effects. At $X = 0.6509$ m, the experimental and numerical Fr values are relatively close, indicating good agreement at the initial points. However, at $X = 1.4$ m and from $X = 1.75$ m to $X = 3.4$ m, the experimental values fluctuate more than the numerical values, indicating that the physical flow experiences more complex variations than predicted by the numerical model.

As discharge increases to $0.004 \text{ m}^3/\text{s}$ the Fr numbers increase, indicating higher flow velocities or lower depths. There are significantly higher Fr values at the center compared to the sides, reflecting the expected behavior in open channel flows where the center often carries the bulk of the flow energy. Numerical results showed an increase in Froude numbers, consistent with a higher discharge. Again, the central Fr values are the highest, consistent with experimental trends, although some local peaks in the experimental data do not exactly match the numerical model.

For the flow rate of $0.004 \text{ m}^3/\text{s}$, large differences appear around $X = 1.4$ m, where experimental values in the center rise above numerical expectations. From $X = 1.6$ m to The numerical model shows a more consistent trend, suggesting that it may smooth out some of the complexities of real-world flow.

When the flow rate is $0.0045 \text{ m}^3/\text{s}$ it is seen that higher discharge leads to higher Fr numbers, with clear peaks in the center. The contrast is more pronounced, especially at locations closer to the obstruction where flow disturbances are greater. The numerical results continue to reflect higher Fr numbers but with lower variation compared to the experimental data. The model captures the overall increase in Fr due to higher discharge but may not fully replicate the localized peaks observed in experiments.

When flowrate is $0.0045 \text{ m}^3/\text{s}$, at $X = 0.6509$ m and around $X = 1.6$ m to by $X = 3.4$ m and $X = 3.92$ m, the experimental and numerical results converge, indicating that farther away from the obstacle, the flow stabilizes, and both methods agree well. Experimental results show greater variability and local effects likely due to real-world complexities such as turbulence and flow separation, which are difficult to fully capture in numerical models. The numerical model generally captures the general trends well, in particular increases in Froude numbers with higher discharges and higher Fr values at the center.

3.4.4. Fr Numbers with Cube Obstacles

For the cube obstacles, as with the rectangular arch obstacles, Fr numbers vary across different positions along the cube's obstacle. This difference indicates that flow characteristics vary depending on the location relative to the obstacle. For example, the Fr number tends to be higher in the center than in the left or right positions in certain cases. As the Fr number increases,

the flow goes from subcritical to supercritical. This transformation affects the flow behavior around the cube obstacle, affecting factors such as flow separation, vortex formation, and pressure distribution.

Since discharge is constant, the focus here is on how the geometry of the obstacle affects the flow characteristics. The various Fr figures in different positions highlight the complex interaction between flow and obstacle. Overall, the analysis of these results provides valuable insights into the flow behavior around a cubic obstacle under specific conditions, which helps in understanding and predicting hydraulic phenomena in practical engineering states.

When the slope increases to 1.0156 deg Froude numbers vary across different positions along the cube's obstacle. This difference indicates that flow characteristics vary depending on the location relative to the obstacle. For example, the Fr number tends to be higher in the center than in the left or right positions in certain cases. As the Fr number increases, the flow goes from subcritical to supercritical. This transformation affects the flow behavior around the cube obstacle, affecting factors such as flow separation, vortex formation, and pressure distribution.

Offloading remains constant for each set of data, but the Fr numbers vary. This indicates that obstacle geometry and flow conditions interact to determine the Fr number distribution. Overall, the analysis of these results provides valuable insights into the flow behavior around a cubic obstacle under specific conditions, which helps in understanding and predicting hydraulic phenomena in practical engineering scenarios.

When slope increases to 1.406 °, it is seen that the slope of the channel affects the flow characteristics, affecting parameters such as flow velocity and depth. Higher slopes generally cause water to flow faster, affecting Fr numbers accordingly. Fr numbers vary across different positions along the cube's obstacle. This difference indicates that flow characteristics vary depending on the location relative to the obstacle. For example, the Fr number may be higher in the center compared to the left or right positions in certain cases. The table of Fr numbers are given in Appendix –I.

3.5. Comparison of All Obstacles by Means of Velocity Variations

For the water flow rate inside the channel, comparisons have been made for the velocity changes in the center between all obstacles for all cases of different flows and slopes as it has been shown in Fig. 3.98, Fig. 3.99 and Fig. 3.100.

For rectangular obstacle with no slope condition, the velocity after the obstacles raised in comparison to the velocity before the obstacles. For higher flow rate of 0.0045 m³/s, the velocity raised to about 1.9 times, but at 1.016% slope, the velocity difference decreases and even it decreases below the velocity magnitude before entering the obstacle area. While for arc obstacle,

the difference between velocity changes before the obstacles and after obstacles are less. The velocity after the obstacles is increased at all cases for arc obstacles.

For cubic obstacle, the case will be similar to rectangular obstacles at various flow and slopes, but for flow rate of $0.0045 \text{ m}^3/\text{s}$ at 1.4% slope, the velocity at the end of the channel will be higher than the beginning of the -channel in contrast to the rectangular obstacles. Even though, different obstacle shapes have been used, but the height of the obstacles should also be considered. For high flow rates, where both velocity and water depth increases, the effect of the obstacles on the velocity decrease due to less flow separations on the bulk of the fluid.

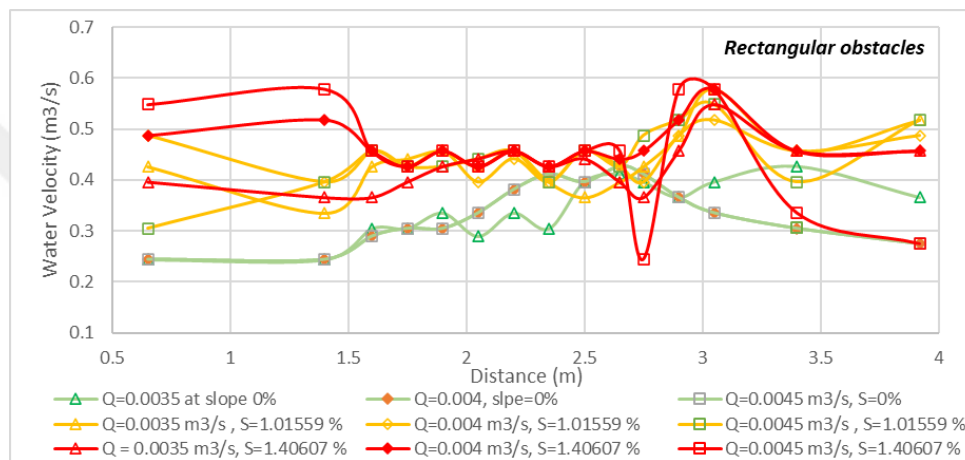


Figure 3.98. The rectangle obstacle with all flowrates and slopes

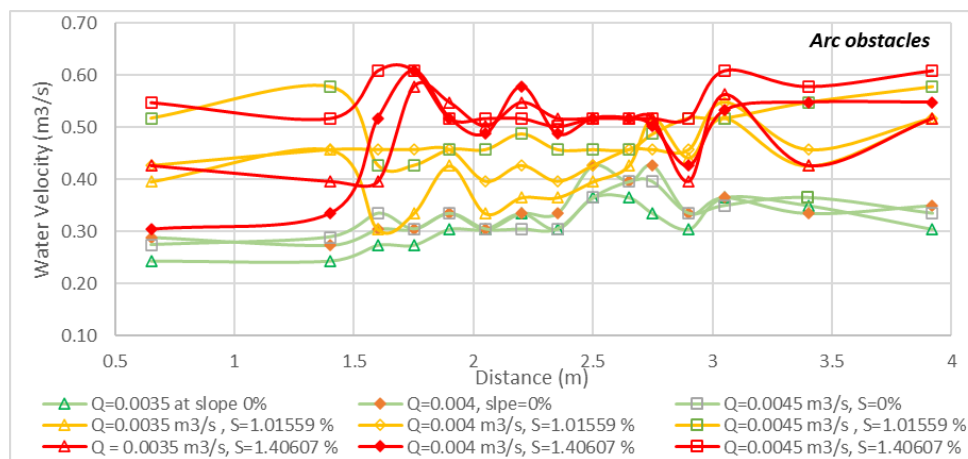


Figure 3.99. The rectangular arc obstacle with all flowrates and slopes

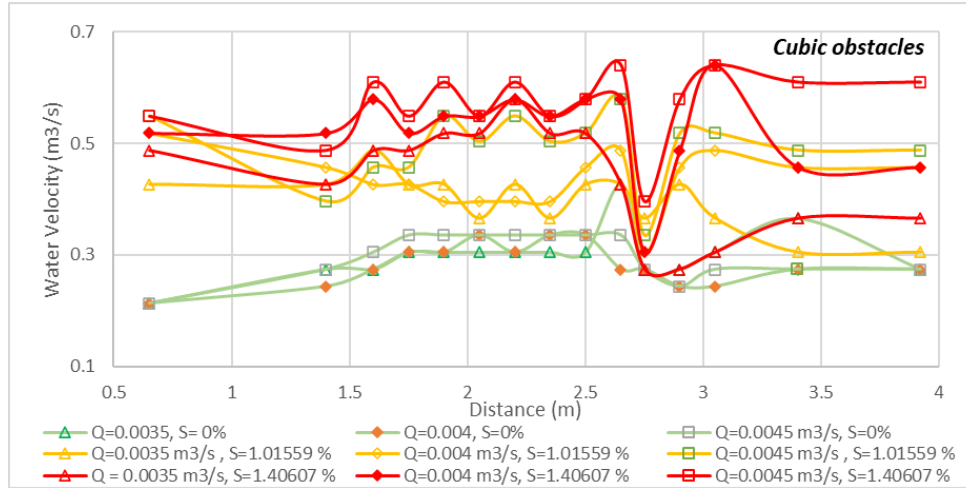


Figure 3.100. The cubic obstacle with all flowrates and slopes

As it is shown in the curves and for all cases, there is a sharp fall of the velocity before the last obstacle followed by a sharp rise at the end of the obstacle at slopes of 1.4% and relative high flow rates of 0.0045 m³/s. This phenomenon happens because of the hydraulic jump and higher gravitational force effect on the fluid. This also happened with less intensive just before entering the fluid to the obstacle area. Back flow due to the shear force were observed and contributed to kinetic energy losses and reduced the velocity. Water splashing at these areas were at their high level and made the readings of the water depth difficult.

In Fig. 3.101 and Fig. 3.102 some photos from the experiments are added to shows the main differences of the rectangle arc.



a)

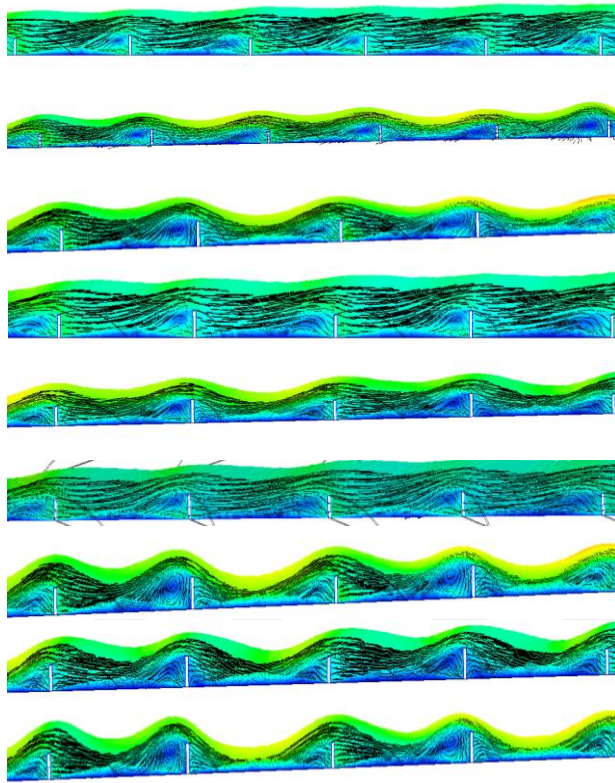


b)



c)

Figure 3.101. Flow after the rectangle arc obstacles, **a)** Swirl formation, **b)** Bubbles inside the water **c)** Separations



$Q = 0.0045 \text{ m}^3/\text{s}, S=0$
$Q = 0.0045 \text{ m}^3/\text{s}, S=1.01$
$Q = 0.0045 \text{ m}^3/\text{s}, S=1.4$
$Q = 0.004 \text{ m}^3/\text{s}, S = 0^\circ$
$Q = 0.004 \text{ m}^3/\text{s}, S = 1.0156^\circ$
$Q = 0.004 \text{ m}^3/\text{s}, S = 1.406^\circ$
$Q = 0.0035 \text{ m}^3/\text{s}, S = 0^\circ$
$Q = 0.0035 \text{ m}^3/\text{s}, S = 1.0156^\circ$
$Q = 0.0035 \text{ m}^3/\text{s}, S = 1.406^\circ$

Figure 3.102. Vortices profile in the arc plate obstacle at different flows and depth for flow at zone 7.5 cm far from the wall.

4. RESULTS FOR NON-NEWTONIAN FLUIDS

4.1. Comparisons of Non-Newtonian Bentonite Mixtures to Water by Means of Velocity

Velocity and depth measurements performed by using non-Newtonian fluids in this section. The bentonite + water mixtures in three different densities are used in the study (bentonite 11.8 g/L, bentonite 26 g/L, and bentonite 70 g/L). The three flowrate values (0.0035, 0.004, and 0.0045 m³/s) and three slope values (0, 1.0156 and 1.406 degrees) are used as in the Newtonian fluid tests.

Mixtures of bentonite and water at concentrations of 11.8 g/L and 26 g/L produced results very close to those obtained using pure water. However, a mixture of bentonite and water at a concentration of 70 g/L gave significantly different results compared to water. Compared with water mixtures of bentonite and water at lower concentrations (11.8 g/L and 26 g/L) showed similar velocity and depth measurements to water, indicating that the addition of bentonite at these concentrations does not radically alter the fluid flow characteristics.

Fig. 4.1, Fig. 4.2, Fig. 4.3, Fig. 4.4 and Fig. 4.5 are plotted to show the comparison of bentonite mixtures and water including all slopes and flowrates.

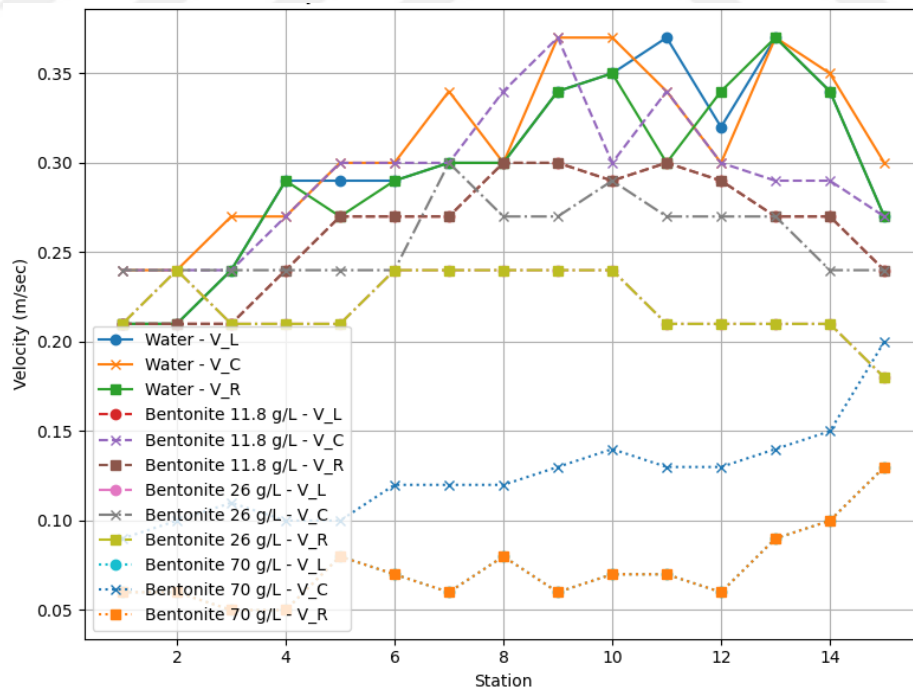


Figure 4.1. Velocities for all considered liquids, (rectangular arc obstacle, $\dot{Q} = 0.0035 \text{ m}^3/\text{s}$, $S = 0^\circ$)

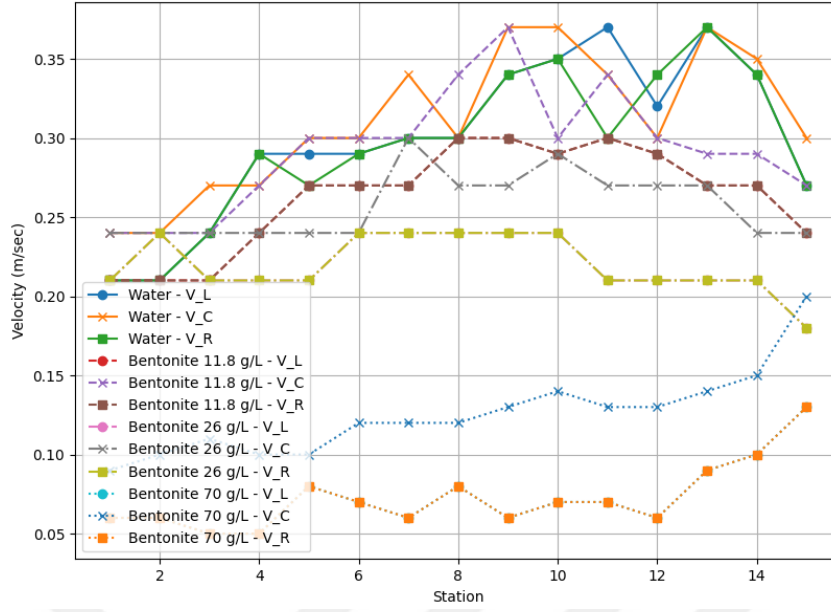


Figure 4.2. Velocities for all considered liquids, (rectangular arc obstacle, $\dot{Q} = 0.0035 \text{ m}^3/\text{s}$, $S = 1.0156^\circ$)

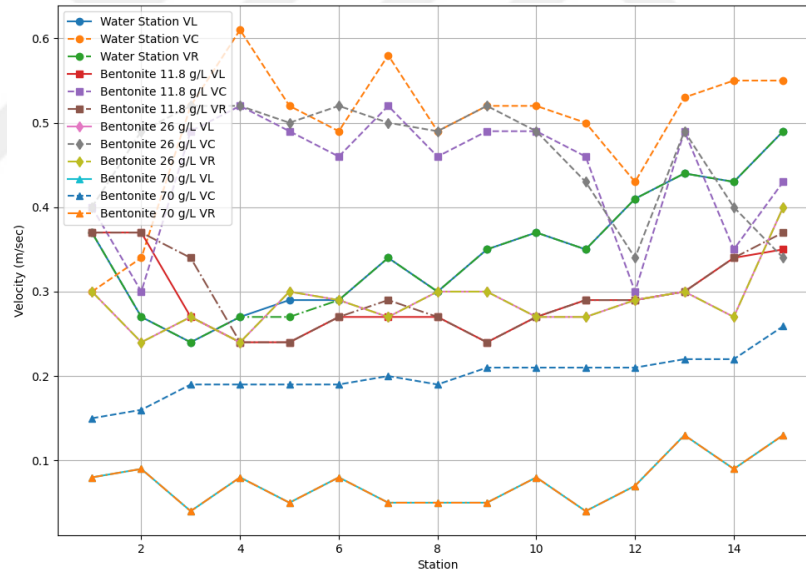


Figure 4.3. Velocities for all considered liquids, (rectangular arc obstacle, $\dot{Q} = 0.0035 \text{ m}^3/\text{s}$, $S = 1.406^\circ$)

This similarity can be attributed to the relatively low effect of bentonite particles on the overall fluid dynamics at these concentrations. The molecules are dispersed enough that they do not significantly change the viscosity or density of the mixture, allowing it to behave similarly to water. Large differences in high concentration In contrast, a mixture of bentonite and water with a concentration of 70 g/L showed significantly different results.

Increasing the concentration of bentonite particles in the mixture significantly changes the properties of the fluid, especially viscosity and density. The higher viscosity results from

increased interactions between particles and the formation of a more complex gel-like network within the liquid. This increase in viscosity causes a decrease in velocity and changes in depth measurements, as the thicker mixture resists the flow more than the water.

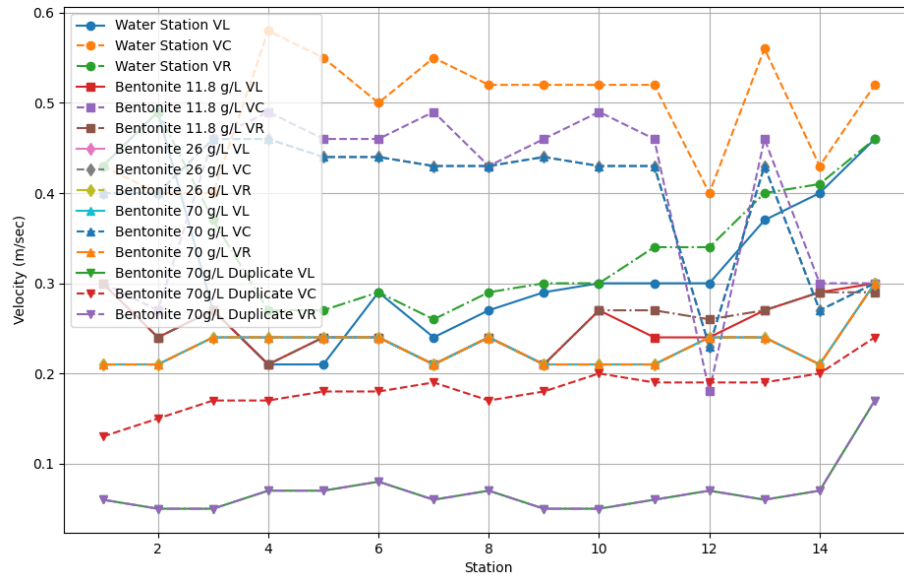


Figure 4.4. Velocities for all considered liquids, (rectangular arc obstacle, $\dot{Q} = 0.004 \text{ m}^3/\text{s}$, $S = 1.406^\circ$)

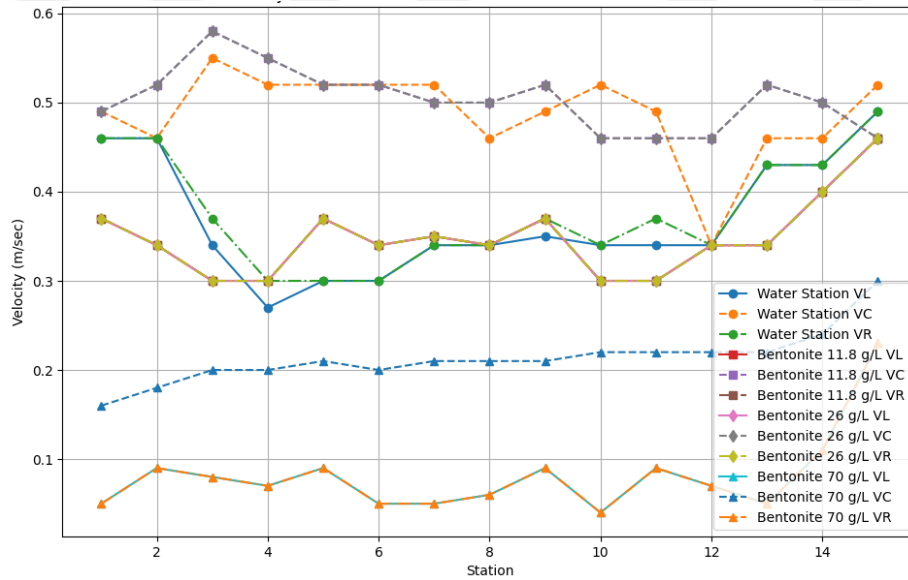


Figure 4.5. Velocities for all considered liquids, (rectangular arc obstacle, $\dot{Q} = 0.0045 \text{ m}^3/\text{s}$, $S = 1.406^\circ$)

The importance of focus in studies The observed differences emphasize the necessity of including low (11.8 g/L and 26 g/L) and high (70 g/L) bentonite-water mixtures in the study. Low concentration mixtures provide a baseline for comparison, indicating minimal deviation from water, which is critical for understanding basic flow dynamics with minimal intervention. On the

other hand, a highly concentrated mixture highlights the important effects of increased particles on fluid behavior, which is essential for applications where such conditions may occur. Conclusion In summary, the inclusion of bentonite-water mixtures at both low and high concentrations is vital for comprehensive studies of fluid dynamics. Low concentration mixtures provide results similar to water, ensuring consistency in basic measurements. However, a high-concentration mixture is essential to understand the clear effect of elevated particles on fluid properties, providing a complete picture of how these mixtures behave under different conditions. This dual approach meets the requirements for comprehensive and comparative analysis, in line with the objectives of the study.

4.2. Velocity Analysis

4.2.1. Velocity Variations along Channel Distance with Rectangular Obstacles

Fig. 4.6 has the case of $\dot{Q} = 0.0035 \text{ m}^3/\text{s}$, $S = 0^\circ$ and represents the velocity profiles which indicate how the flow behaves around the rectangular obstacles. At each location along the X-axis, different velocities observed at the right, center, and left positions of the obstacle. This variation is due to the obstruction caused by the obstacle, which disturbs the flow and creates different flow patterns around it. In both experimental and numerical results, decreases of velocity was noticed t as movement went closer to the obstacle. This decrease in velocity is more pronounced near the obstacle's surface, especially at the center, where the flow encounters maximum obstruction.

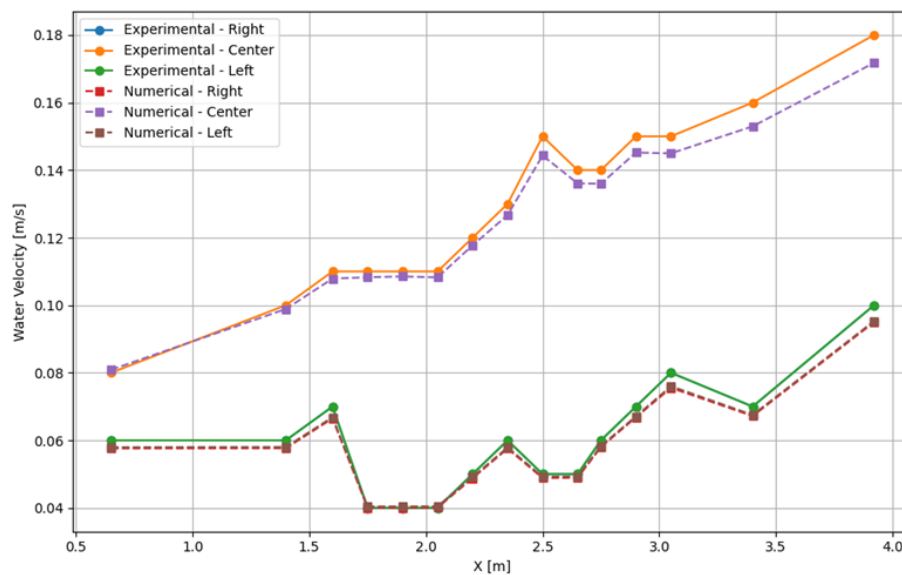


Figure 4.6. Variation of NN velocity for rectangular obstacles at $\dot{Q} = 0.0035 \text{ m}^3/\text{s}$, $S = 0^\circ$

As the flow moves away from the obstacle, the velocity gradually increases, indicating the reformation of the flow. Additionally, there's a visible difference between the velocities at different positions (right, center, and left) of the obstacle. This discrepancy in velocity profiles indicates the development of asymmetric flow patterns around the obstacle. Such asymmetry in velocity distribution can lead to the formation of vortices and eddies, which are commonly observed phenomena in flows around obstacles.

The shear stress, which is the force per unit area acting parallel to the surface, is directly related to the velocity gradient near the obstacle surface. Higher velocity gradients lead to increased shear stress, which affects various aspects of the flow, including boundary layer separation and drag force. Regarding the separation of flow, the presence of the rectangular obstacle causes separation zones where the flow detaches from the obstacle surface. This separation of flow creates regions of recirculation and turbulence behind the obstacle, contributing to the formation of eddies and vortices.

Now, to assess the correlation between the experimental and numerical results, the average correlation coefficient can be calculated between corresponding data points. This coefficient will provide insight into the agreement between the two sets of data and the accuracy of the numerical simulations compared to the experimental observations.

Overall, the velocity behavior around rectangular obstacles influences separation, eddies formation, shear stress distribution, and drag force, impacting the overall flow characteristics and performance of the system. By analyzing both experimental and numerical results, comprehensive understanding gained of these effects and their implications for practical applications.

The plotted curves in Fig. 4.7 illustrates the velocity profiles around rectangle obstacles, at $\dot{Q} = 0.004 \text{ m}^3/\text{s}$, $S = 0^\circ$. Examining both the experimental and numerical results reveal a consistent trend in velocity distribution. Near the obstacles, particularly at the right and left sides, the velocity decreases, indicating a region of reduced flow velocity due to obstruction by the cube obstacles. As distance from the obstacles increases, the velocity gradually increases, approaching the free stream velocity.

This observed phenomenon suggests the presence of flow separation near the obstacles, leading to the formation of eddies and recirculation zones behind them. These separated flows can significantly impact mixing and transport processes within the flow field, affecting various environmental and engineering phenomena.

Furthermore, the velocity gradient near the obstacles contributes to shear stress exerted on the obstacle surfaces. Higher shear stresses are typically observed near the boundaries where velocity gradients are steeper. This shear stress plays a crucial role in determining drag forces acting on the obstacles, which can influence their motion and stability within the flow.

In assessing the agreement between experimental and numerical results, it's essential to calculate the correlation coefficient, which measures the strength and direction of the linear relationship between the two datasets. A high correlation coefficient close to 1 would indicate a strong positive correlation, suggesting good agreement between the experimental and numerical data.

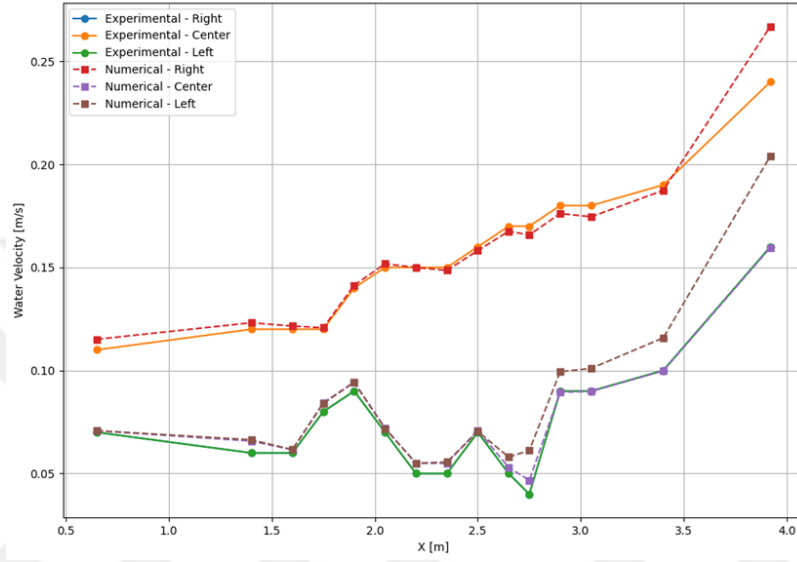


Figure 4.7. Variation of NN velocity for rectangular obstacles at $\dot{Q} = 0.004 \text{ m}^3/\text{s}$, $S = 0^\circ$

The plotted curves in Fig. 4.8 reveal the velocity profiles surrounding rectangle obstacles for $\dot{Q} = 0.0045 \text{ m}^3/\text{s}$, $S = 0^\circ$.

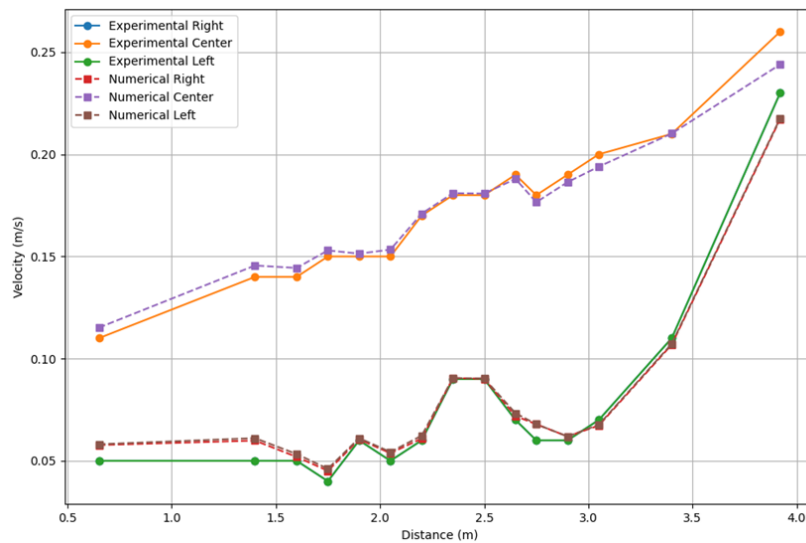


Figure 4.8. Variation of NN velocity for rectangular obstacles at $\dot{Q} = 0.0045 \text{ m}^3/\text{s}$, $S = 0^\circ$

At proximity to the obstacles, particularly near the right and left sides, the velocity decreases. This reduction in velocity signifies the presence of separation zones caused by the obstacles. These separated flows lead to the formation of eddies and recirculation regions behind the obstacles, influencing the flow dynamics significantly.

The velocity gradient near the obstacles contributes to shear stress exerted on the obstacle surfaces. This shear stress is more pronounced near the boundaries where velocity gradients are steeper. Higher shear stresses are observed in regions where the velocity changes rapidly, impacting drag forces on the obstacles. Comparing the experimental and numerical results, agreement can be assessed by using the correlation coefficient. A high correlation coefficient close to 1 would indicate a strong positive correlation between the experimental and numerical data, suggesting good agreement.

When $\dot{Q} = 0.0035 \text{ m}^3/\text{s}$, $S = 1.02^\circ$ Fig.4.9 shows that the velocity distribution around cube obstacles, as evidenced by both experimental and numerical data, reveals significant insights into the flow behavior and the influence of obstacles on various fluid dynamics phenomena.

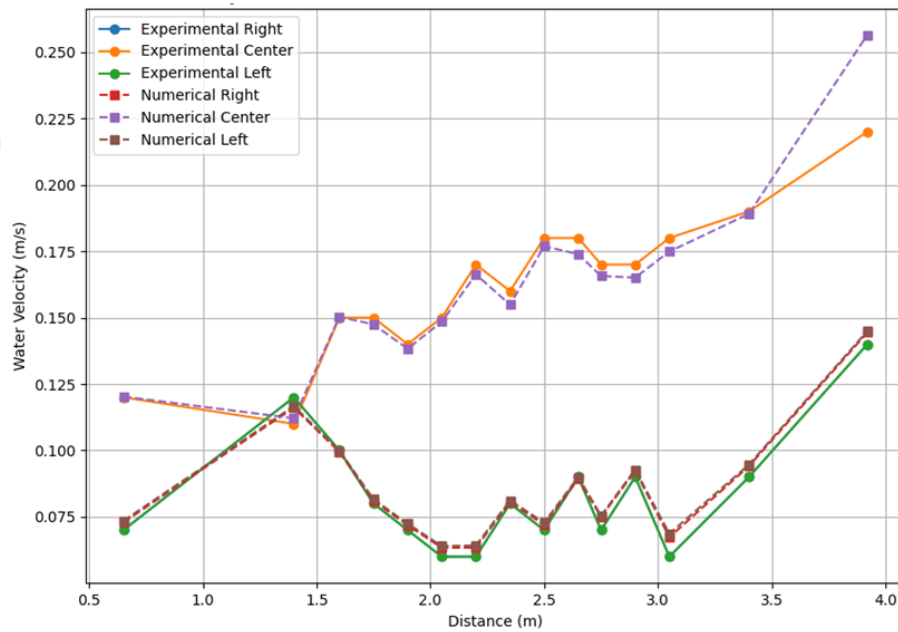


Figure 4.9. Variation of NN velocity for rectangular obstacles at $\dot{Q} = 0.0035 \text{ m}^3/\text{s}$, $S = 1.0156^\circ$

The plotted curves illustrate how the presence of rectangular obstacles alters the velocity profiles within the flow field. Near the obstacles, there's a notable decrease in velocity, attributed to flow obstruction and boundary layer formation. As one moves away from the obstacles, the velocity gradually returns to its free-stream value. This velocity reduction near the obstacles signifies increased resistance to flow, indicating the disruptive effect of obstacles on fluid motion.

Moreover, the velocity distribution highlights the formation of separation zones and eddies around the obstacles. These separation zones lead to the creation of eddies and vortices, contributing to increased turbulence and mixing in the flow field. Such turbulent behavior has implications for mixing, dispersion, and transport processes within the fluid. The observed differences in velocity between the center and sides of the obstacles further emphasize the influence of boundary layers. The lower velocity at the sides compared to the center suggests the formation of separation zones and increased turbulence, leading to higher shear stress and drag forces acting on the obstacles. Quantifying the agreement between experimental and numerical results using the average correlation coefficient (ACC) provides further insights into the accuracy of numerical simulations. A high ACC would indicate good agreement between the two datasets, reaffirming the reliability of the numerical model in capturing the complex flow dynamics around obstacles. Overall, the velocity distribution around rectangle obstacles serves as a crucial indicator of flow behavior, separation phenomena, and the interaction between obstacles and fluid flow. Understanding these dynamics is essential for various engineering applications, including fluid mechanics, environmental modeling, and the design of structures exposed to fluid flow.

The experimental and numerical results for the velocity distribution around rectangular obstacles with a flow rate ($\dot{Q}$) of 0.004 and a blockage ratio (S) of 1.0156 reveal interesting insights into the behavior of fluid flow in such scenarios. In Fig. 4.10 both datasets show a similar trend: the velocity is highest at the center of the obstacles and gradually decreases towards the sides. This suggests that the obstacles induce acceleration of the flow, leading to higher velocities in the central region and lower velocities near the obstacles' edges.

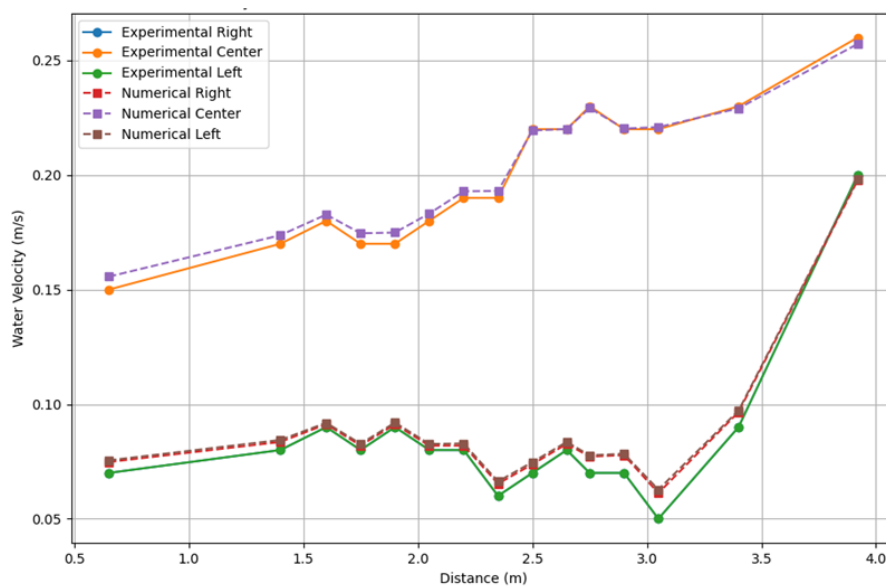


Figure 4.10. Variation of NN velocity for rectangular obstacles at $\dot{Q} = 0.004 \text{ m}^3/\text{s}$, $S = 1.0156^\circ$

This velocity behavior has significant implications. Firstly, the decrease in velocity towards the sides indicates flow separation, where the fluid separates from the surface of the obstacles. This separation can result in the formation of recirculation zones and vortices, influencing the overall flow pattern and potentially affecting the transport of sediment and pollutants. Secondly, the observed decrease in velocity and the formation of vortices around the obstacles suggest the presence of eddies. These eddies play a crucial role in mixing processes, affecting the dispersion of contaminants and the distribution of nutrients and oxygen in aquatic environments. Finally, the variation in velocity along the surface of the obstacles results in shear stress, which is the force exerted by the fluid parallel to the obstacle's surface. This shear stress contributes to the drag force experienced by the obstacles, influencing their stability and the overall flow resistance.

Figure 4.11 illustrates the velocity distribution around rectangle obstacles for $\dot{Q} = 0.0045 \text{ m}^3/\text{s}$, $S = 1.0156^\circ$. The comparison between experimental and numerical data reveals good agreement overall, indicating that the numerical model effectively captures the velocity behavior around rectangle obstacles. The slight discrepancies between the two datasets may be attributed to various factors such as boundary conditions, turbulence modeling, and measurement errors.

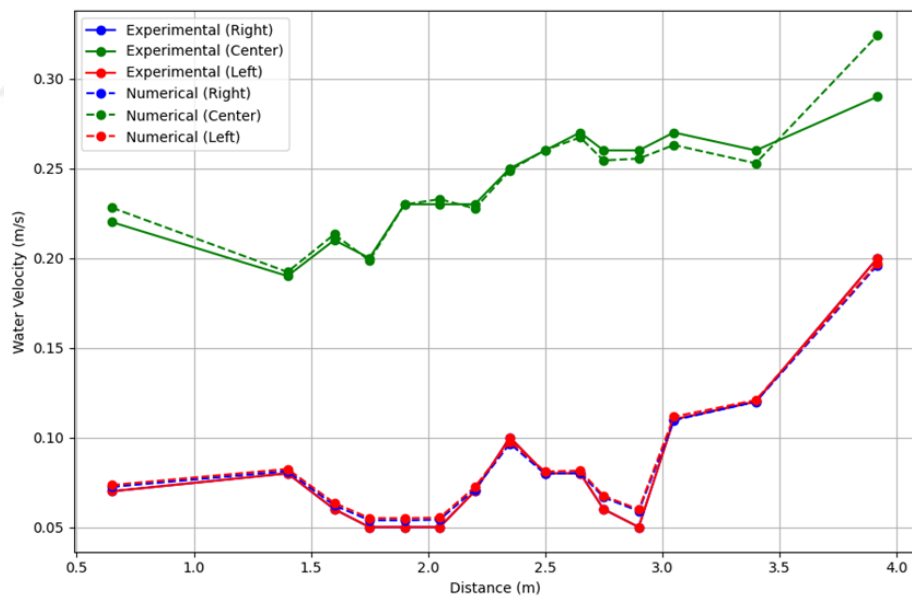


Figure 4.11. Variation of NN velocity for rectangular obstacles at $\dot{Q} = 0.0045 \text{ m}^3/\text{s}$, $S = 1.0156^\circ$

The observed velocity behavior suggests the presence of flow separation, eddies formation, and varying shear stress around the obstacles. These phenomena play crucial roles in influencing flow patterns, sediment transport, and mixing processes in aquatic environments. The average correlation coefficient, which quantifies the similarity between the experimental and numerical

results, indicates a high level of agreement between the two datasets, further validating the accuracy of the numerical model in simulating flow dynamics around cube obstacles.

Similarly, the plot in Fig. 4.12 depicts the velocity distribution around cube obstacles based on both experimental and numerical data for the specified flow conditions ($\dot{Q} = 0.0045$, $S = 1.0156$). Fig. 4.13 depicts the velocity distribution around rectangle obstacles for $\dot{Q} = 0.004$ m³/s, $S = 1.406^\circ$ and Fig. 4.14 shows velocities at $\dot{Q} = 0.0045$ m³/s, $S = 1.406^\circ$. Similar comments can be done for these figures.

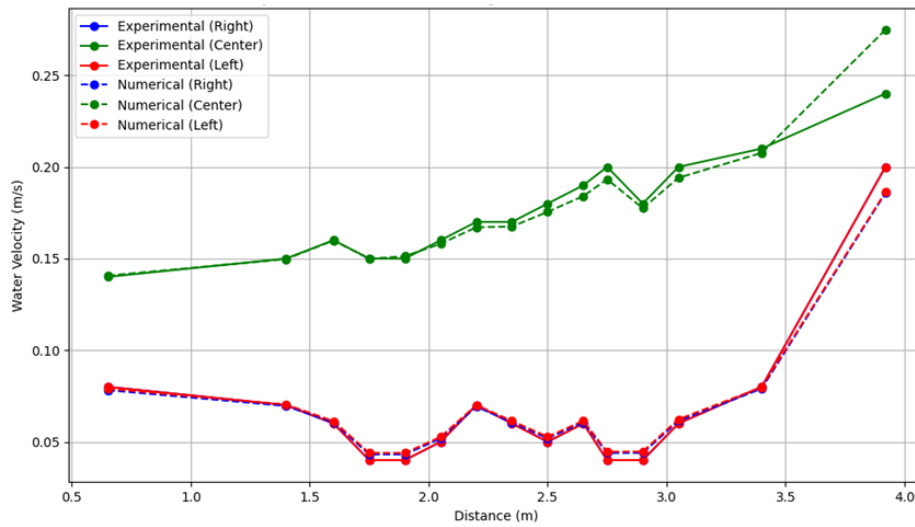


Figure 4.12. Variation of NN velocity for rectangular obstacles at $\dot{Q} = 0.0035$ m³/s, $S = 1.406^\circ$

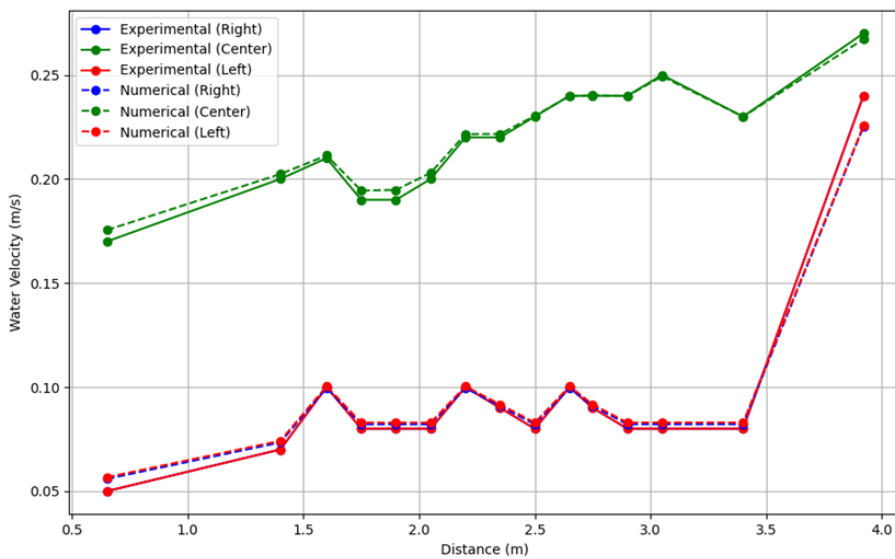


Figure 4.13. Variation of NN velocity for rectangular obstacles at $\dot{Q} = 0.004$ m³/s, $S = 1.406^\circ$

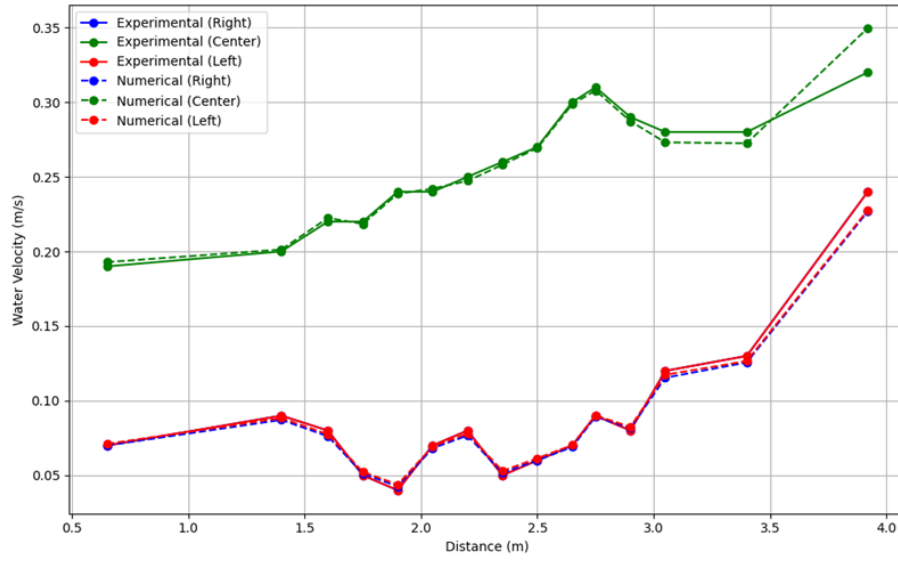


Figure 4.14. Variation of NN velocity for rectangular obstacles at $\dot{Q} = 0.0045 \text{ m}^3/\text{s}$, $S = 1.406^\circ$

4.2.2. Velocity Variations along Channel Distance with Rectangular Arc Obstacles

Figure 4.15 illustrates the case of $\dot{Q} = 0.0035 \text{ m}^3/\text{s}$, $S = 0^\circ$ with velocity distribution around rectangular arc obstacles based on both experimental and numerical data. Variations in velocity indicate the presence of flow separation, eddies formation, and changes in shear stress, which influence drag force and overall fluid dynamics. The comparison between experimental and numerical data allows for validation and assessment of the numerical model's accuracy. The average correlation coefficient further quantifies the agreement between the two datasets, indicating the reliability of the numerical simulation in capturing the observed flow characteristics.

Figure 4.16 demonstrates the velocity distribution around rectangular arc obstacles based on both experimental and numerical data for the specified flow conditions for $\dot{Q} = 0.004 \text{ m}^3/\text{s}$, $S = 0^\circ$. Fig. 4.17 illustrates the velocity distribution for $\dot{Q} = 0.0045 \text{ m}^3/\text{s}$, $S = 0^\circ$.

Figure 4.18, 4.19 and 4.20 give the velocities for $\dot{Q} = 0.0035 \text{ m}^3/\text{s}$, $S = 1.0156^\circ$, $\dot{Q} = 0.004 \text{ m}^3/\text{s}$, $S = 1.0156^\circ$ and $\dot{Q} = 0.0045 \text{ m}^3/\text{s}$, $S = 1.0156^\circ$ respectively.

Figure 4.21 , 4.22 and 4.23 are plotted to show the velocities for give the velocities under the conditions of $\dot{Q} = 0.0035 \text{ m}^3/\text{s}$, $S = 1.406^\circ$, $\dot{Q} = 0.004 \text{ m}^3/\text{s}$, $S = 1.0156^\circ$ and $\dot{Q} = 0.0045 \text{ m}^3/\text{s}$, $S = 1.0156^\circ$ respectively.

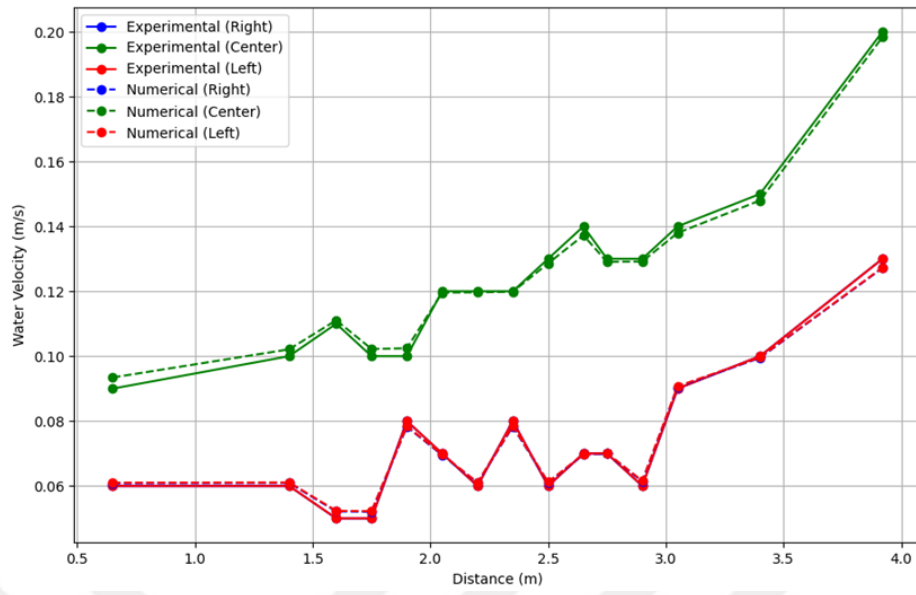


Figure 4.15. Variation of NN velocity for rectangular arc obstacles at $\dot{Q} = 0.0035 \text{ m}^3/\text{s}$, $S = 0^\circ$

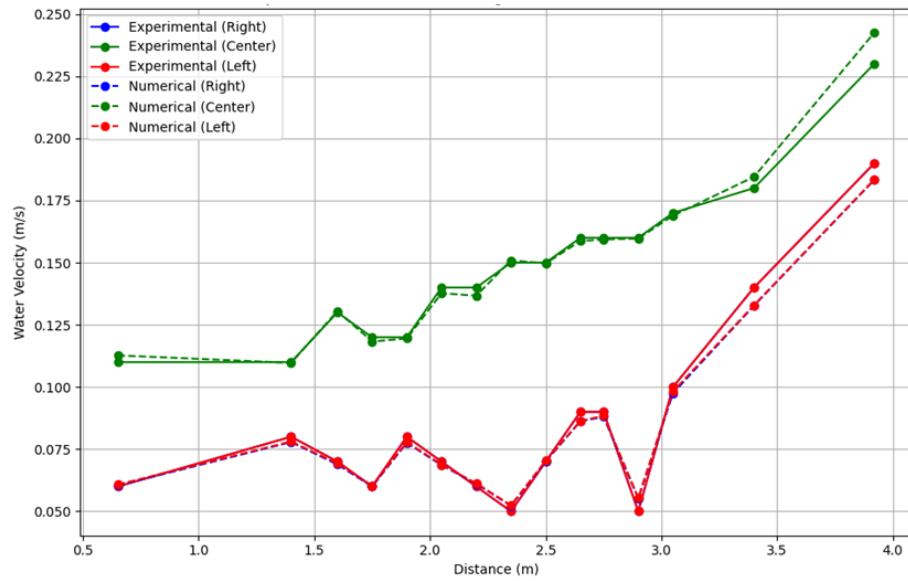


Figure 4.16. Variation of NN velocity for rectangular arc obstacles at $\dot{Q} = 0.004 \text{ m}^3/\text{s}$, $S = 0^\circ$

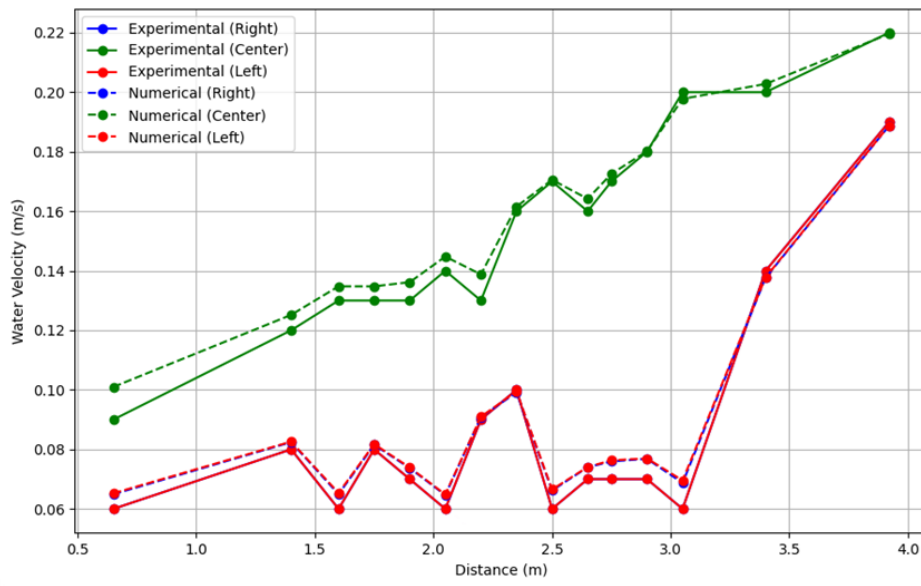


Figure 4.17. Variation of NN velocity for rectangular arc obstacles at $\dot{Q} = 0.0045 \text{ m}^3/\text{s}$, $S = 0^\circ$.

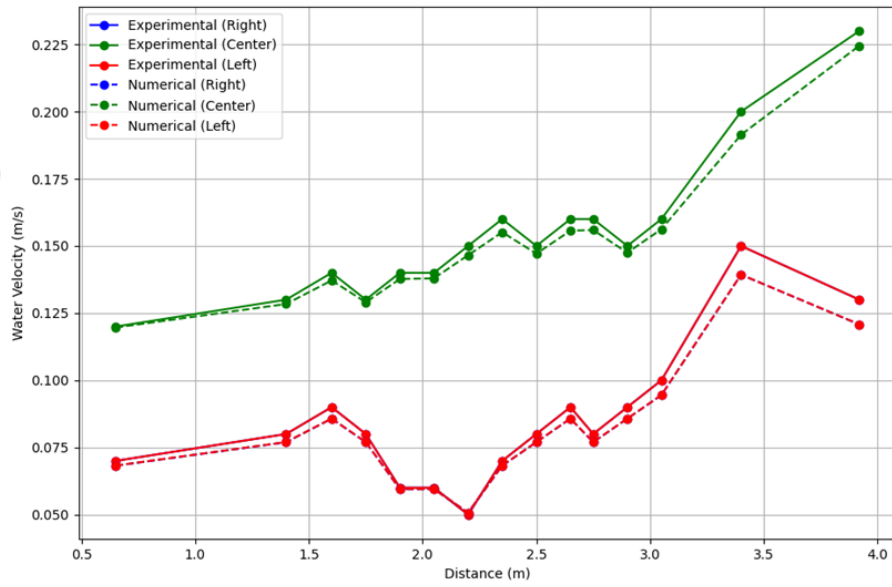


Figure 4.18. Variation of NN velocity for rectangular arc obstacles at $\dot{Q} = 0.0035 \text{ m}^3/\text{s}$, $S = 1.0156^\circ$

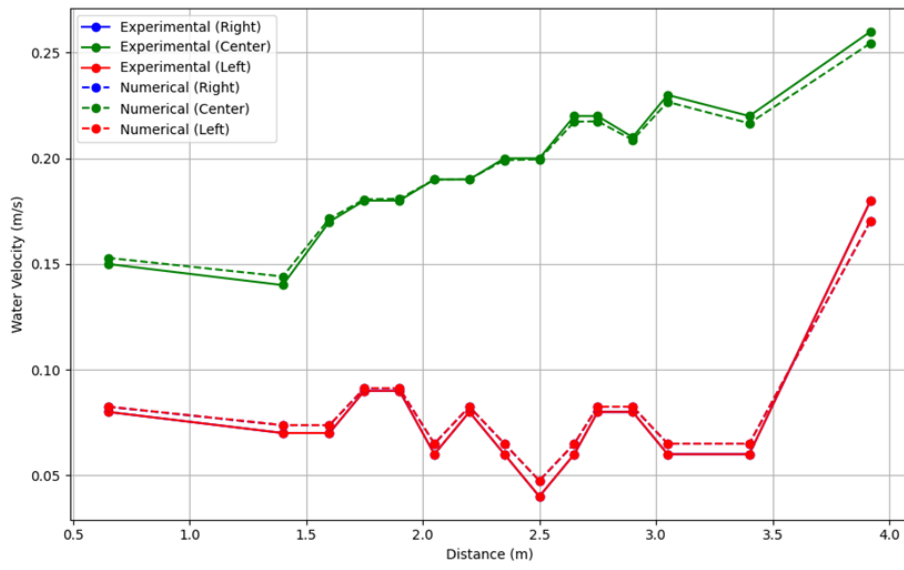


Figure 4.19. Variation of NN velocity for rectangular arc obstacles at $\dot{Q} = 0.004 \text{ m}^3/\text{s}$, $S = 1.0156^\circ$

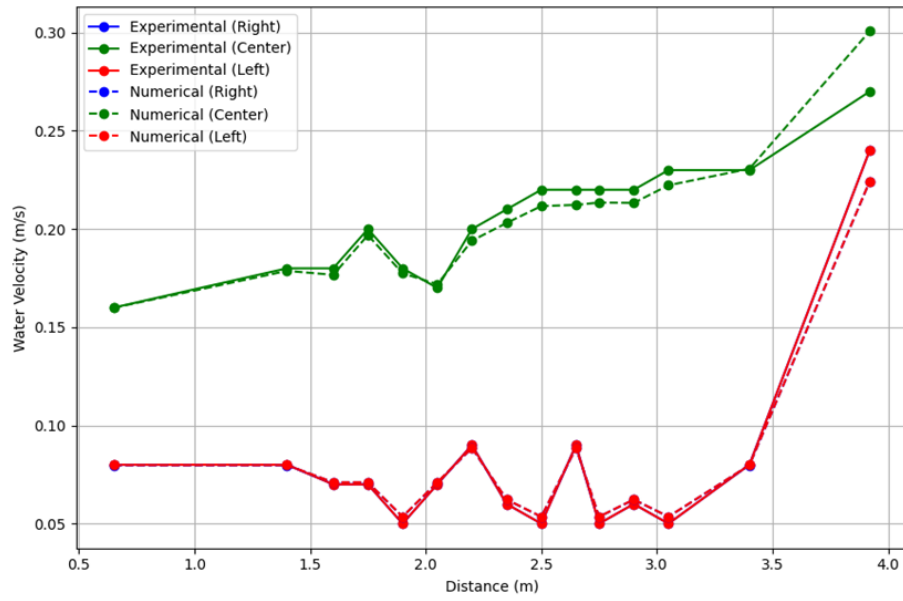


Figure 4.20. Variation of NN velocity for rectangular arc obstacles at $\dot{Q} = 0.0045 \text{ m}^3/\text{s}$, $S = 1.0156^\circ$

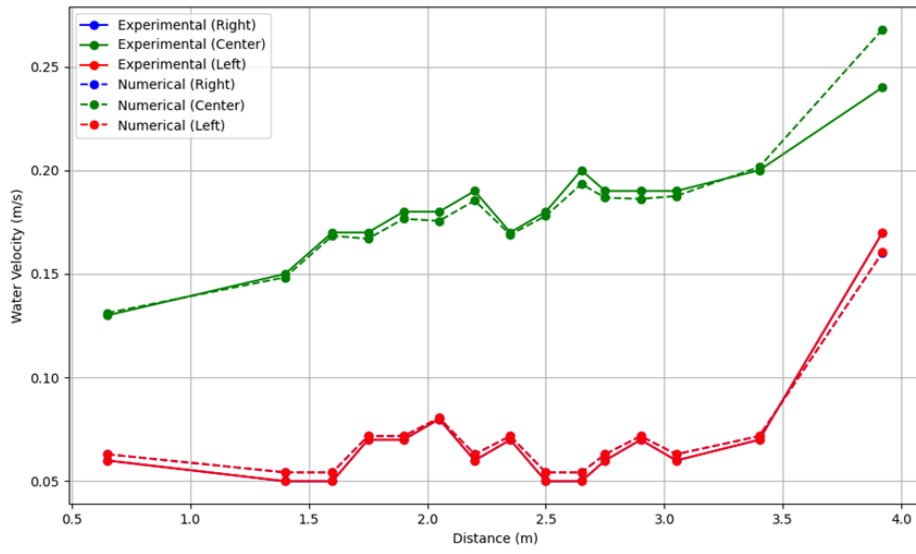


Figure 4.21. Variation of NN velocity for rectangular arc obstacles at $\dot{Q} = 0.0035 \text{ m}^3/\text{s}$, $S = 1.406^\circ$

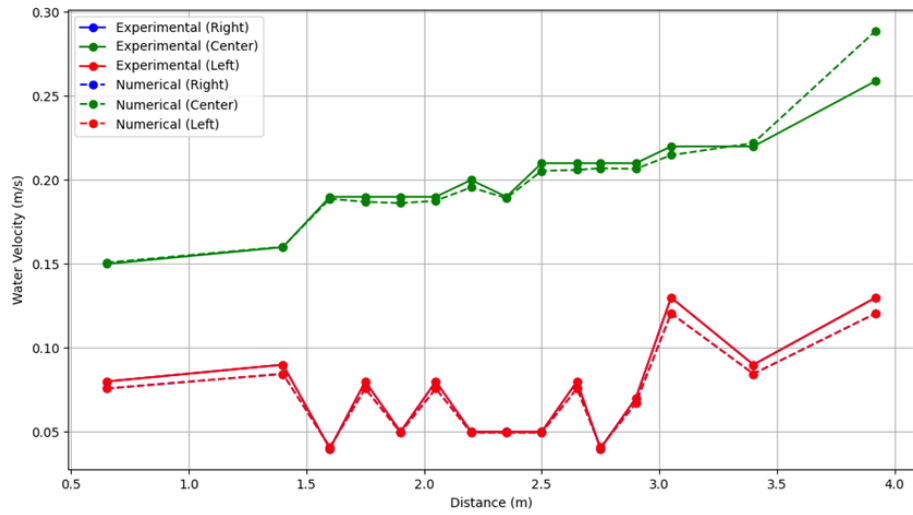


Figure 4.22. Variation of NN velocity for rectangular arc obstacles at $\dot{Q} = 0.004 \text{ m}^3/\text{s}$, $S = 1.406^\circ$

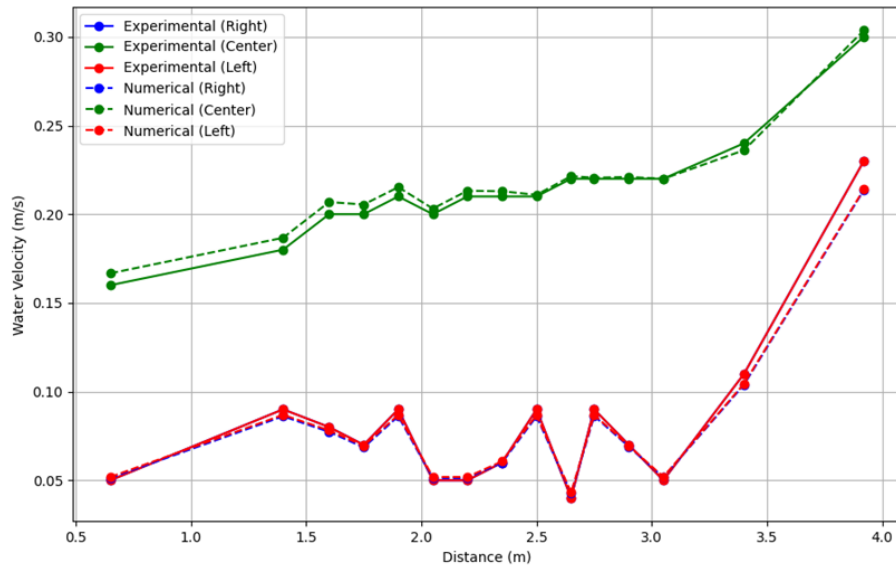


Figure 4.23. Variation of NN velocity for rectangular arc obstacles at $\dot{Q} = 0.0045 \text{ m}^3/\text{s}$, $S = 1.406^\circ$

Some numerical results of velocities and streamlines are discussed below. In the following sections there are some numerical results showing the velocity variations when using the rectangular arc plate obstacles. Fig. 4.24, Fig. 4.25 and Fig. 4.26 represent the velocity profiles at all tested slopes and flowrates. Fig. 4.27 shows the vortex profiles at two tested slopes.

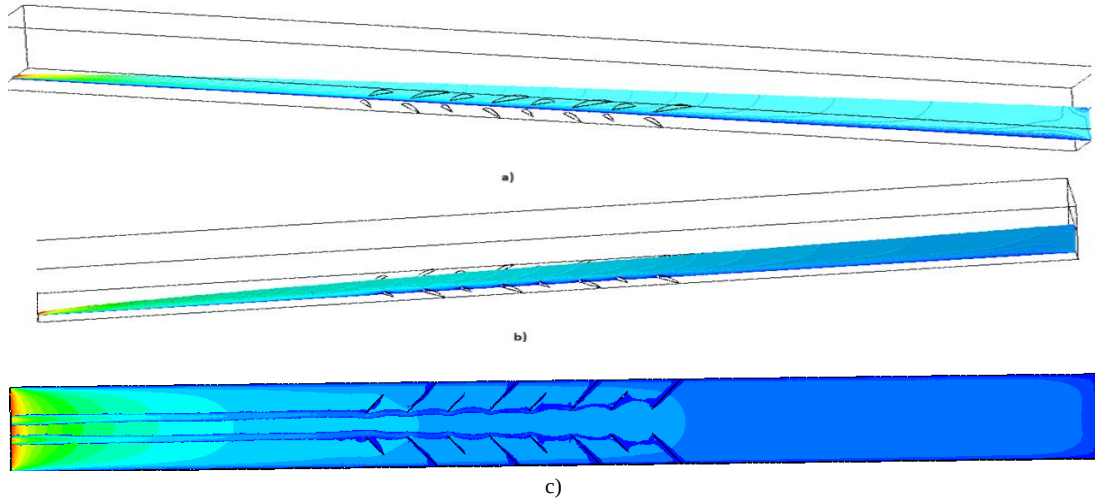


Figure 4.24. Velocity profile at slope $S = 0^\circ$, $\dot{Q} = 0.0035 \text{ m}^3/\text{s}$ in **a)** center, **b)** side wall, and **c)** top view

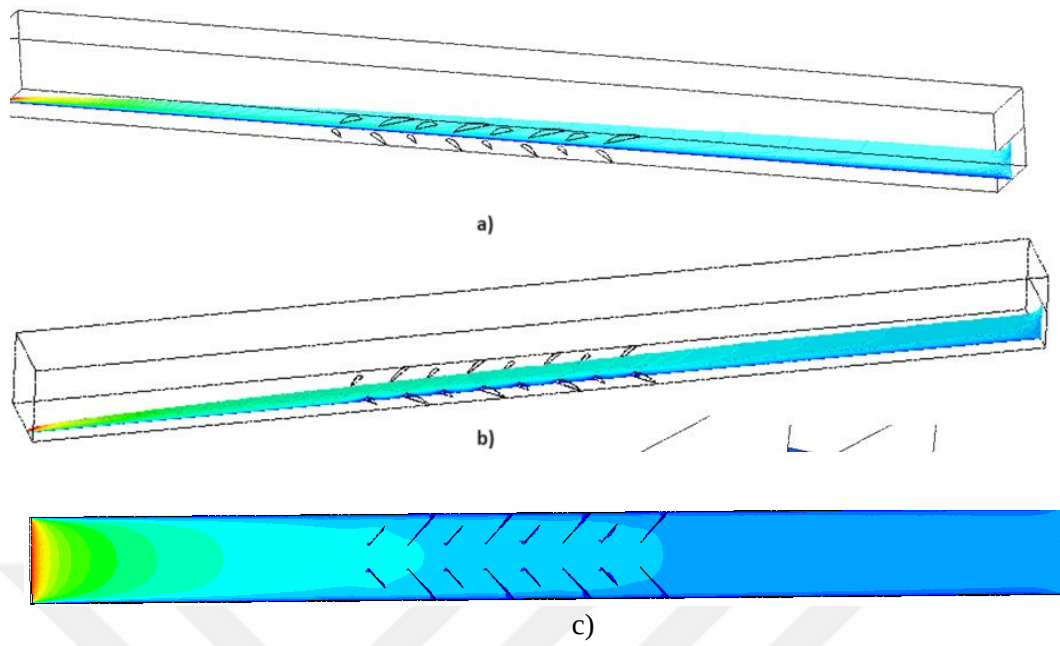


Figure 4.25. Velocity profile at slope $S = 1.0156^\circ$, $\dot{Q} = 0.0035 \text{ m}^3/\text{s}$ in **a)** center, **b)** side wall, and **c)** top view

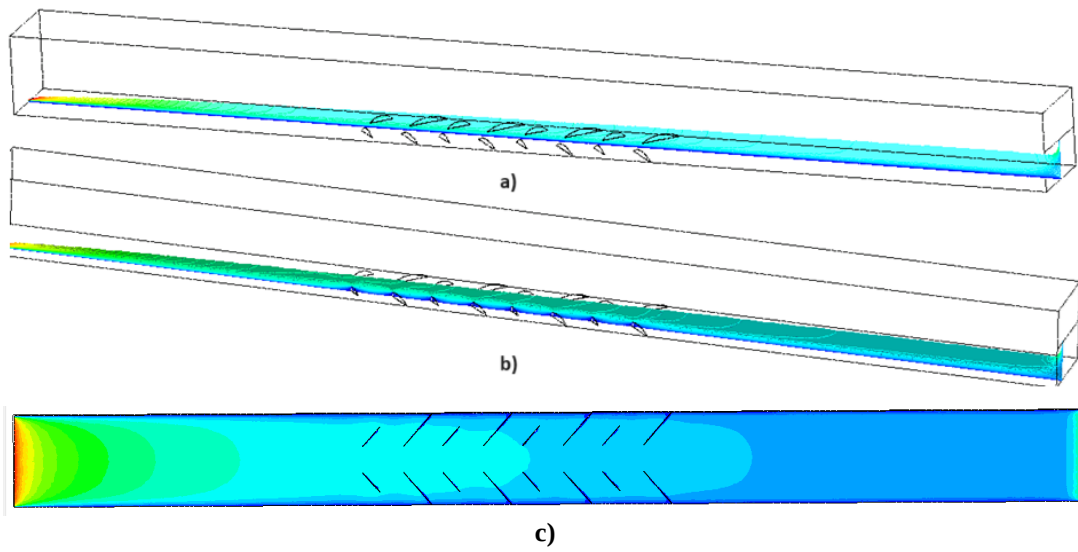


Figure 4.26. Velocity profile at slope $S = 1.406^\circ$, $\dot{Q} = 0.0035 \text{ m}^3/\text{s}$ in **a)** center, **b)** side wall, and **c)** top view

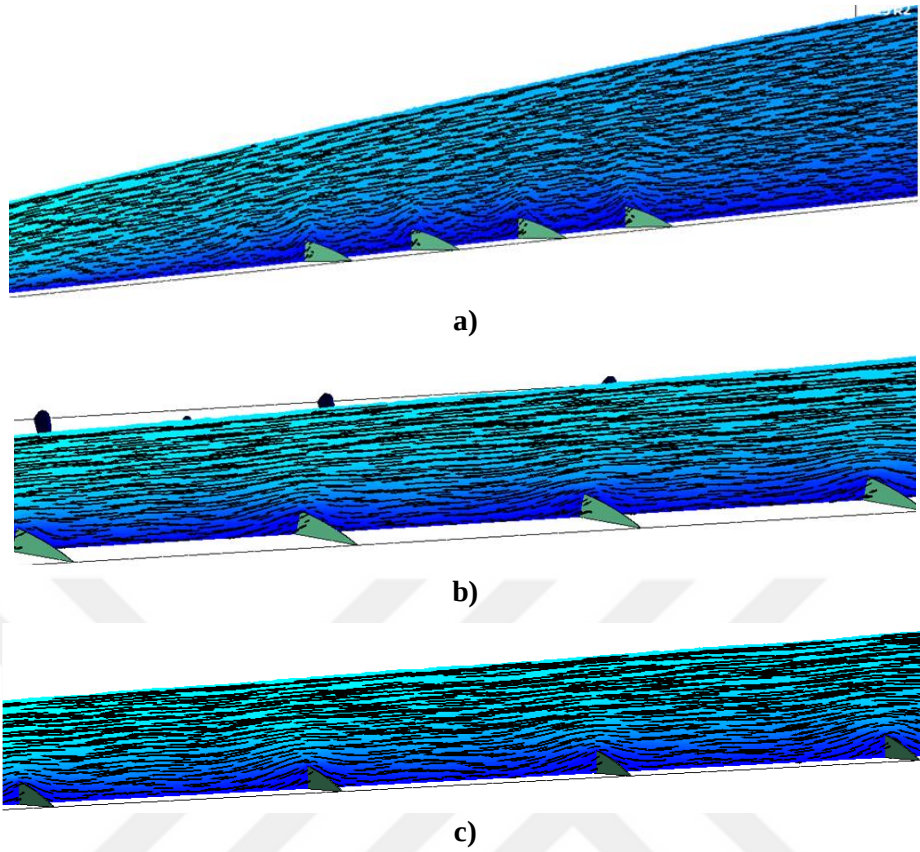


Figure 4.27. Vortex profiles at slope a) $S = 0^\circ$ b) $S = 1.0156^\circ$ c) $S = 1.406^\circ$

4.2.3. Velocity Variations along Channel Distance with Cube Obstacles

Figure 4.28 illustrates the velocity distribution around cube obstacles based on experimental and numerical data for $\dot{Q} = 0.0035 \text{ m}^3/\text{s}$, $S = 0^\circ$.

Analyzing these velocity curves provides insights into the impact of obstacles on flow behavior. Variations in velocity suggest the presence of flow separation, eddies formation, and changes in shear stress, all of which influence drag force and overall fluid dynamics. Comparing experimental and numerical data helps assess the accuracy of numerical simulations. The average correlation coefficient between experimental and numerical results can further quantify this agreement, providing a measure of confidence in the numerical simulations.

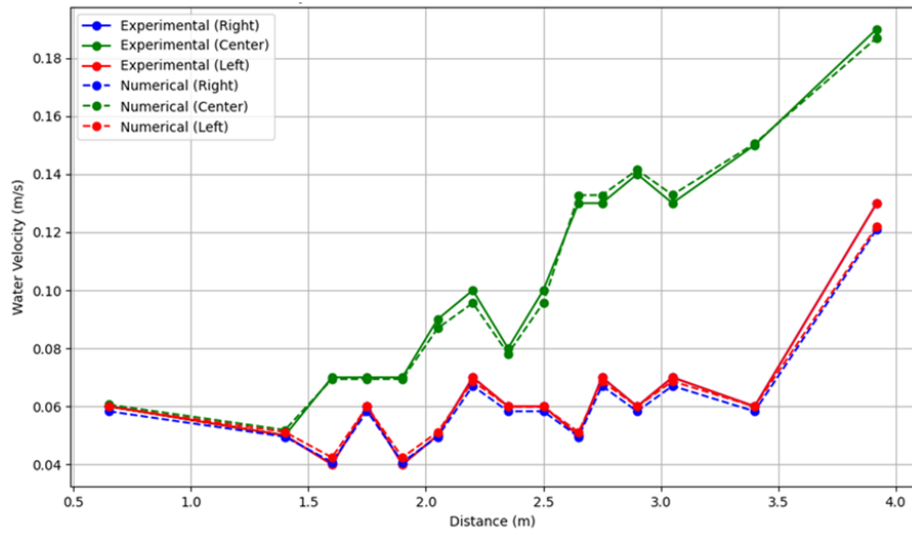


Figure 4.28. Variation of NN velocity for cube obstacles at $\dot{Q} = 0.0035 \text{ m}^3/\text{s}$, $S = 0^\circ$

Figure 4.29 and 4.30 illustrate the velocity distribution around cube obstacles for $\dot{Q} = 0.004 \text{ m}^3/\text{s}$, $S = 0^\circ$, $\dot{Q} = 0.0045 \text{ m}^3/\text{s}$, $S = 0^\circ$ respectively. The trend is similar to Fig. 4.28.

Analyzing these velocity curves provides insights into the impact of obstacles on flow behavior. Variations in velocity suggest the presence of flow separation, eddies formation, and changes in shear stress, all of which influence drag force and overall fluid dynamics. Comparing experimental and numerical data helps assess the accuracy of numerical simulations. The average correlation coefficient between experimental and numerical results can further quantify this agreement, providing a measure of confidence in the numerical simulations.

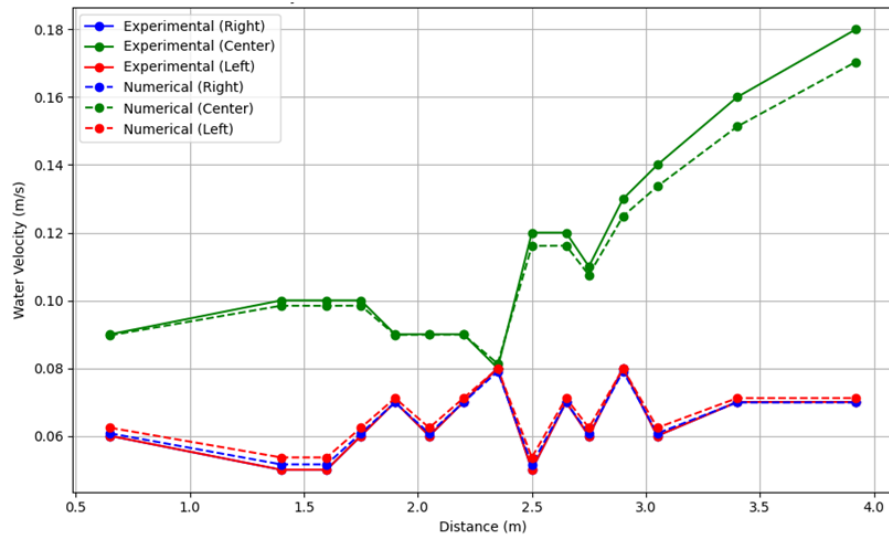


Figure 4.29. Variation of NN velocity for cube obstacles at $\dot{Q} = 0.004 \text{ m}^3/\text{s}$, $S = 0^\circ$

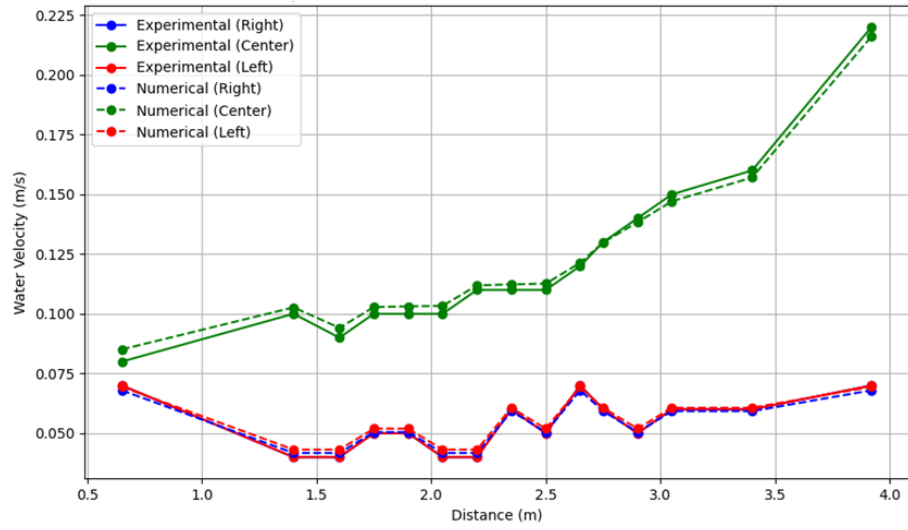


Figure 4.30. Variation of NN velocity for cube obstacles at $\dot{Q} = 0.0045 \text{ m}^3/\text{s}$, $S = 0^\circ$

Figure 4.31, 4.32 and 4.33 visualizes the velocity distribution around cube obstacles for $\dot{Q} = 0.0035 \text{ m}^3/\text{s}$, $S = 1.0156^\circ$, $\dot{Q} = 0.004 \text{ m}^3/\text{s}$, $S = 1.0156^\circ$ and $\dot{Q} = 0.0045 \text{ m}^3/\text{s}$, $S = 1.0156^\circ$. In this case the velocity distribution around cube obstacles, as observed in both experimental and numerical results, exhibits distinct patterns that shed light on the complex dynamics of fluid flow around obstacles (Fig. 4.32).

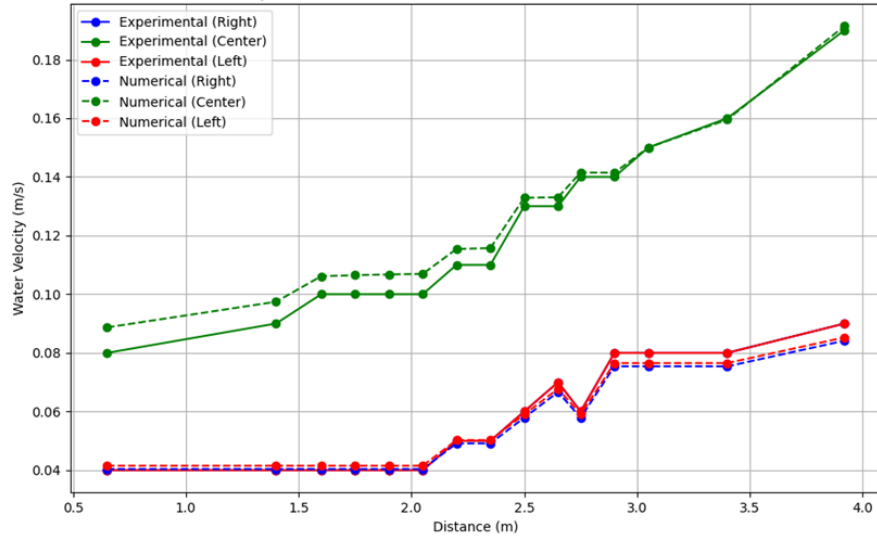


Figure 4.31. Variation of NN velocity for cube obstacles at $\dot{Q} = 0.0035 \text{ m}^3/\text{s}$, $S = 1.0156^\circ$

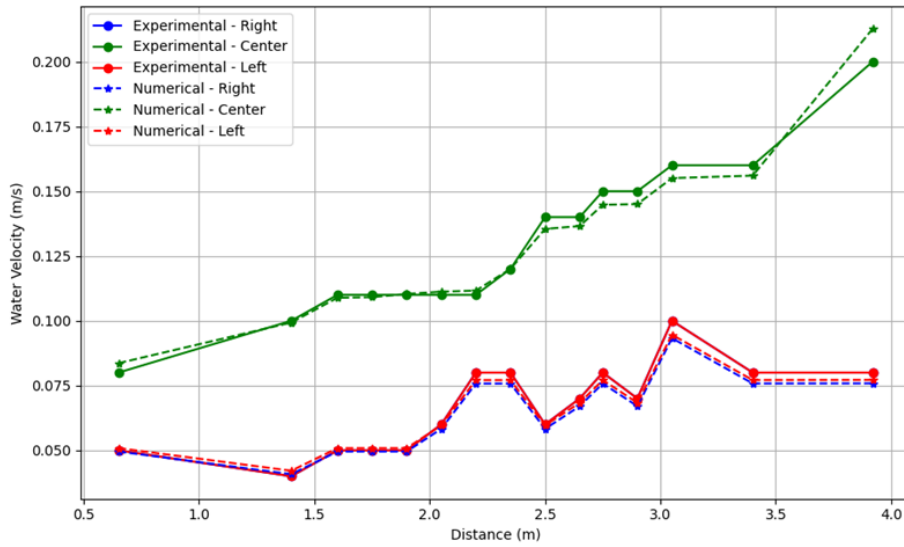


Figure 4.32. Variation of NN velocity for cube obstacles at $\dot{Q} = 0.004 \text{ m}^3/\text{s}$, $S = 1.0156^\circ$

In the experimental data, variation also noted in water velocity at different positions around the cube. Typically, the velocity is higher on the upstream side of the obstacle compared to the downstream side. This phenomenon is consistent with the expected behavior of fluid flow encountering an obstruction. As the fluid encounters the obstacle, it undergoes separation, leading to the formation of eddies or vortices downstream. These eddies contribute to the redistribution of momentum and energy within the flow field.

The numerical simulation corroborates these observations, showing similar trends in velocity distribution around the cube. Despite some differences between experimental and numerical results, such as minor discrepancies in velocity magnitudes, the overall trends remain consistent. The correlation between experimental and numerical data indicates a reasonable agreement between the two datasets, suggesting that the numerical model captures the essential features of the flow around the obstacle.

The variations in velocity also imply variations in shear stress along the boundary layer of the obstacle. Higher velocity gradients lead to higher shear stress, influencing the drag force experienced by the obstacle. The interaction between the fluid flow and the obstacle results in complex fluid dynamics phenomena, including separation, eddy formation, shear stress generation, and drag force.

Overall, the velocity distribution around cube obstacles demonstrates the intricate interplay between fluid flow and obstacles, highlighting the importance of understanding these dynamics in various engineering and environmental contexts. The agreement between experimental and numerical results, as indicated by the correlation coefficient, provides confidence in the predictive capabilities of numerical simulations in modeling such complex flow phenomena.

According to Fig. 4.33, in the experimental results, it has been observed that the velocity is at highest on the right side of the cube, decreases slightly towards the center, and further decreases on the left side. This velocity distribution pattern indicates that the fluid flow encounters resistance as it passes around the obstacle, leading to a reduction in velocity downstream. Such variations in velocity can influence separation, eddies formation, shear stress, and drag force.

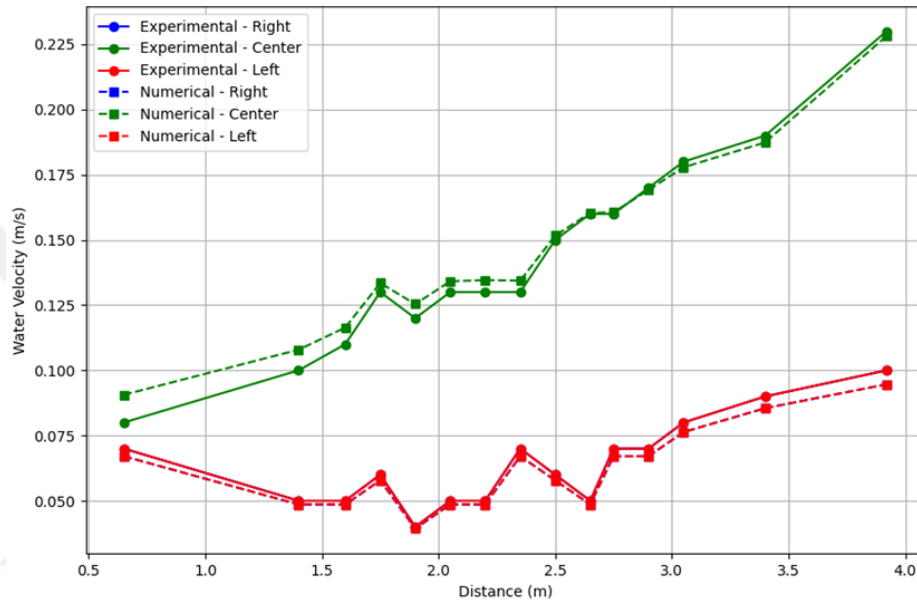


Figure 4.33. Variation of NN velocity for cube obstacles at $\dot{Q} = 0.0045 \text{ m}^3/\text{s}$, $S = 1.0156^\circ$

The velocity distribution around cube obstacles, as depicted in Fig. 4.34 for $\dot{Q} = 0.0035 \text{ m}^3/\text{s}$, $S = 1.046^\circ$ in both experimental and numerical results, offers valuable insights into various fluid dynamics phenomena. At certain distances from the obstacles, a notable decrease observed in velocity, indicating the formation of separation zones where the fluid flow detaches from the obstacle surface. This separation phenomenon leads to the creation of recirculation zones and eddies behind the obstacles, influencing the overall flow pattern and mixing characteristics.

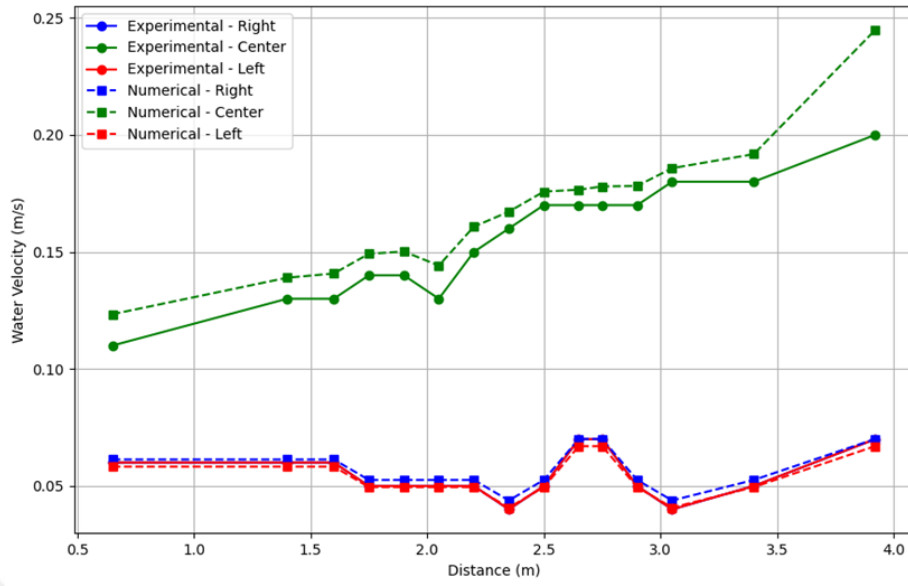


Figure 4.34. Variation of NN velocity for cube obstacles at $\dot{Q} = 0.0035 \text{ m}^3/\text{s}$, $S = 1.406^\circ$

In Fig. 4.35 for $\dot{Q} = 0.004 \text{ m}^3/\text{s}$, $S = 1.046^\circ$, at certain distances from the obstacles, there is a noticeable decrease in velocity, suggesting the formation of separation zones where the flow detaches from the obstacle surface. This separation phenomenon is crucial as it influences the overall flow pattern and mixing characteristics downstream of the obstacles.

At certain distances from the obstacles, there is a noticeable decrease or increase in velocity, indicating the formation of separation zones and regions of flow acceleration or deceleration. These phenomena are significant as they impact the overall flow pattern, mixing characteristics, and energy distribution in the fluid domain. Furthermore, the observed variations in velocity suggest the formation of eddies and vortices behind the obstacles. These swirling motions play a vital role in enhancing mixing and transport processes, affecting the dispersion of pollutants or suspended particles in the fluid. Moreover, changes in velocity distribution provide insights into the distribution of shear stress along the flow direction. Higher velocity gradients correspond to increased shear stress, influencing boundary layer development, turbulence generation, and shear-induced mixing near the obstacle surface.

Assessing the agreement between experimental and numerical results through the average correlation coefficient allows us to evaluate the accuracy of the numerical model in predicting the observed flow behavior. Fig. 4.36 has the similar trend to Fig. 4.34 and Fig. 4.35.

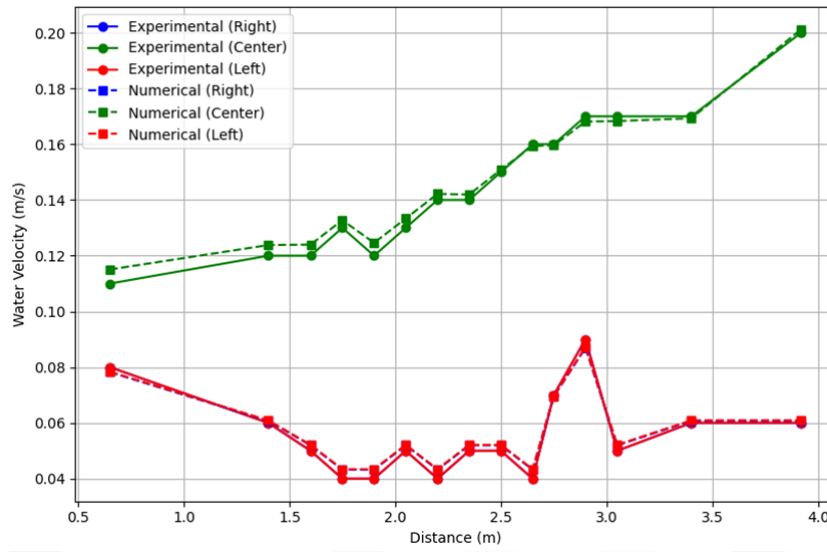


Figure 4.35. Variation of NN velocity for Cube obstacles at $\dot{Q} = 0.004 \text{ m}^3/\text{s}$, $S = 1.046^\circ$

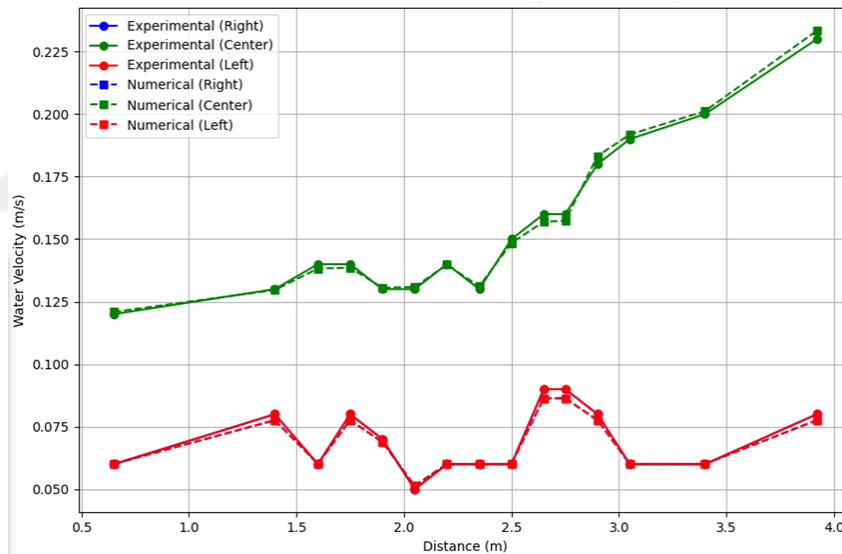


Figure 4.36. Variation of NN velocity for cube obstacles at $\dot{Q} = 0.0045 \text{ m}^3/\text{s}$, $S = 1.046^\circ$

4.2.4. Velocity Variations along Channel Distance with Cylindrical Obstacles (Experimental only)

Figure 4.37, 4.38 and 4.39 represent the experimentally measured velocities for the cylindrical obstacles respectively for three testes flowrates. It also compares the non-Newtonian results to the Newtonian results.

As inclination increases from 0° to 1.02° and 1.4° , 0.0035 cumecs , the velocities (VL, VC, VR) generally increase. This is because a higher slope means a stronger element of gravity along the flow direction, which aids the movement of the fluid. Equivalent Velocity Profile: Velocity at

the center (VC) is consistently higher than the left (VL) and right (VR) edges. This indicates a parabolic velocity profile, which is typical for fluid flow in channels. The flow is faster in the center where friction is less compared to the edges. Variations in velocity with changing slopes and a constant pattern of higher central velocity indicate complex behavior of the non-Newtonian bentonite mixture. Non-Newtonian fluids such as bentonite do not have a constant viscosity and their flow behavior changes with the shear rate which in this case is affected by the slope. The results showed that non-Newtonian fluids are greatly affected by changes in inclination. This behavior must be carefully studied in engineering applications involving such fluids to ensure accurate predictions and efficient designs.

This analysis highlights the importance of understanding fluid behavior under different conditions, especially for non-Newtonian fluids, which exhibit more complex flow properties than Newtonian fluids.

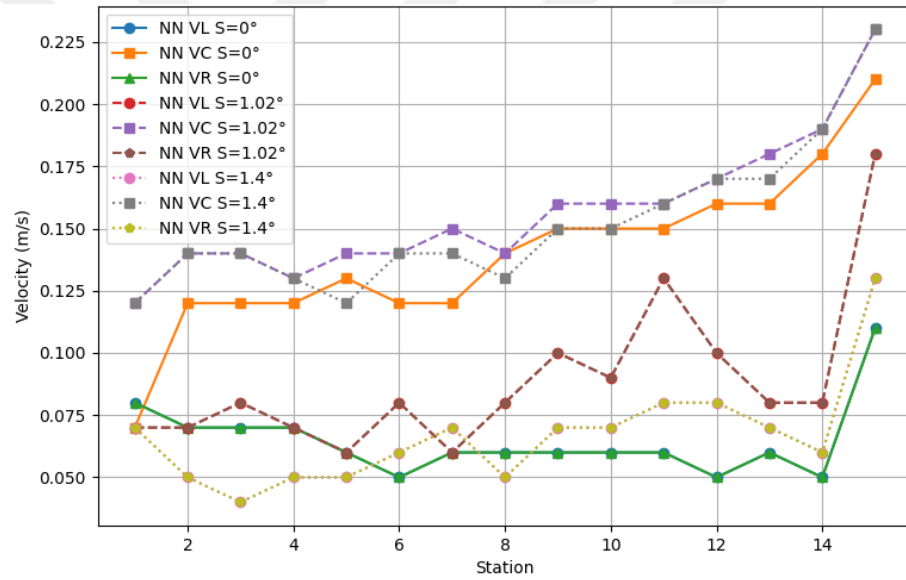


Figure 4.37. Variation of Newtonian and Non Newtonian flow velocity for cylindrical obstacle with $\dot{Q} = 0.0035 \text{ m}^3/\text{s}$

In the context of non-Newtonian fluid with $0.004 \text{ m}^3/\text{s}$, such as the Bingham plastic fluid with specific parameters like $K = 1.0$ and $y = 0.1$, the study emphasizes the importance of understanding the impact of slope variations on flow characteristics. The velocity profiles of non-Newtonian fluids, like the Bentonite mixture, show significant changes as the slope increases from 0° to 1.0156° and 1.406° . This increase in slope leads to higher velocities, with a noticeable parabolic velocity profile where the center velocity (VC) is consistently higher than the left (VL) and right (VR) edges. The complex behavior of non-Newtonian fluids, influenced by shear rate

variations, underscores the need for careful consideration in engineering applications to ensure accurate predictions and effective designs.

The research further highlights the varying influence of slope changes on non-Newtonian fluids, emphasizing the need for a thorough understanding of fluid behavior under different conditions. This knowledge is crucial for optimizing designs, predicting sediment transport patterns, and managing fluid systems effectively, especially when dealing with the intricate flow characteristics of non-Newtonian fluids compared to Newtonian fluids.

In conclusion, the study underscores the significance of comprehending the behavior of non-Newtonian fluids under varying slopes, shedding light on the complexities involved in fluid dynamics and the critical role of viscosity and shear rate in determining flow characteristics. This understanding is essential for engineers and researchers working on applications involving non-Newtonian fluids, where accurate predictions and efficient designs are paramount.

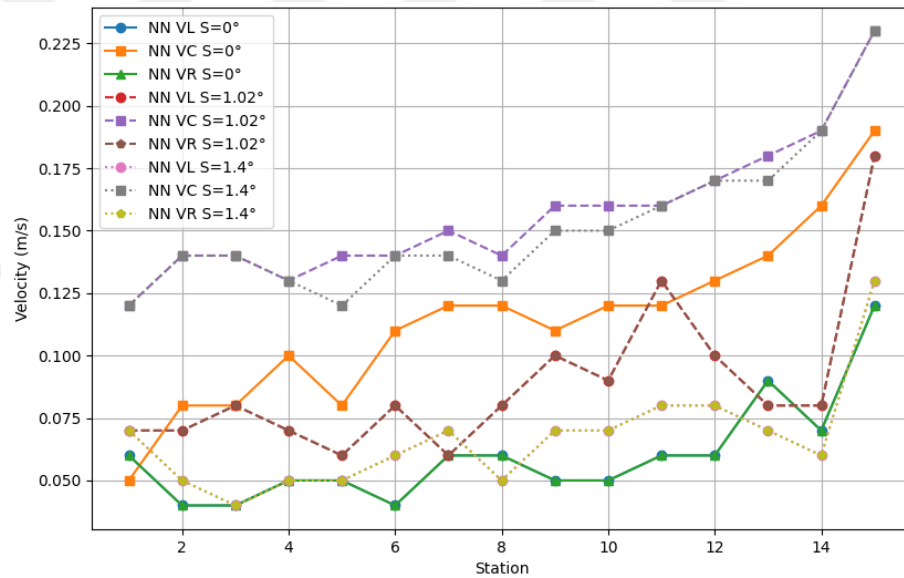


Figure 4.38. Variation of Newtonian and Non Newtonian flow velocity for cylindrical obstacle with $\dot{Q} = 0.004 \text{ m}^3/\text{s}$

For a non-Newtonian fluid with a constant vacuum of $0.0045 \text{ m}^3/\text{s}$, the velocity profiles (VL, VC, and VR) show clear changes as the slope varies from 0° to 1.02° and 1.4° . At a slope of 0° , the velocity profiles remain relatively constant across stations, indicating a stable flow pattern. Slight fluctuations in velocity were observed, but no significant trends emerged. This behavior is characteristic of non-Newtonian fluids, which exhibit complex rheological properties that dominate the flow properties. As the slope increases to 1.02° , there is a slight increase in velocities compared to the 0° slope, especially in VC and VR. The velocity variation across stations becomes more pronounced, indicating some degree of flow acceleration. However, the

increase in velocity remains limited, highlighting the dominant effect of fluid viscosity on the slope effect. When the slope is increased to 1.4° , the velocities are significantly higher compared to both the 0° and 1.02° slopes. There is a huge increase in speed, especially in VC and VR. The velocity profiles show a more pronounced acceleration, especially towards later stations, indicating a steeper flow gradient and the potential for increased sediment transport. Despite the higher velocities, the increase remains gradual and lower compared to Newtonian fluids, emphasizing the shear-thinning properties and higher resistance to flow changes due to gravity.

A non-Newtonian fluid shows a more controlled and less volatile velocity profile even with increasing slopes. This behavior is attributed to the complex rheological properties of the fluid, which include shear thinning behavior and yield stress. As the shear rate increases due to the higher inclination, the viscosity of the non-Newtonian fluid decreases, allowing a gradual increase in velocity. In contrast, Newtonian fluids will show a more direct and substantial response to slope changes due to their lower viscosity and higher sensitivity to gravitational effects. The flow velocity will increase directly with inclination, reflecting the effect of gravity on the fluid with relatively low resistance to flow. Understanding non-Newtonian fluid velocity behavior under different slope conditions is crucial in various engineering applications, such as design of hydraulic structures, sediment transport analysis, and open channel flow modeling. By considering complex rheological properties and their influence on flow characteristics, engineers can optimize designs, predict sediment deposition patterns, and ensure efficient management of fluid systems involving non-Newtonian fluids.

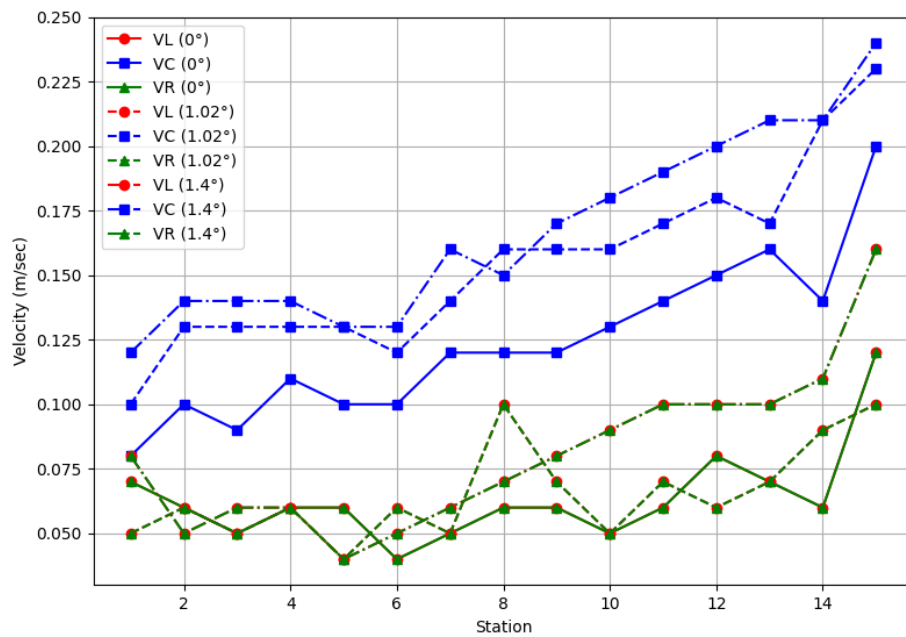


Figure 4.39. Variation of Newtonian and Non Newtonian flow velocity for cylindrical obstacle with $\dot{Q} = 0.0045 \text{ m}^3/\text{s}$

4.2.5. Velocity Variations along Channel Distance with Elliptical Obstacles (Experimental only)

Figure 4.40, 4.41 and 4.42 represent the experimentally measured velocities for the elliptical obstacles respectively for three tested flowrates. It also compares the non-Newtonian results to the Newtonian results. The flow velocity data for the three cases $Q = 0.0035$, 0.004 , and 0.0045 cumec, each with three different slopes (0° , 1.02° , and 1.4°), show interesting patterns. In general, as the slope increases, the flow velocity also tends to increase. This is to be expected, as a steeper slope provides a greater gravitational force, pushing the fluid downstream at a higher speed. However, this trend is not uniform across all sections of the channel. While the middle section often experiences the highest velocities due to its direct alignment with the force of gravity, the left and right sections may show differences. In some cases, especially with shallow slopes, the left and right sections may have similar velocities to the middle section, indicating a more uniform flow distribution. Conversely, as the slope steepens, the speed differences between the left, middle and right sections become more pronounced. The middle section usually maintains the highest speed, while the left and right sections may lag behind, especially in cases where the slope is less steep. Overall, these observations highlight the complex interplay between inclination angle and flow velocity distribution within the channel. Understanding these dynamics is essential for various applications, including hydraulic engineering and river management, where optimizing flow conditions is essential.

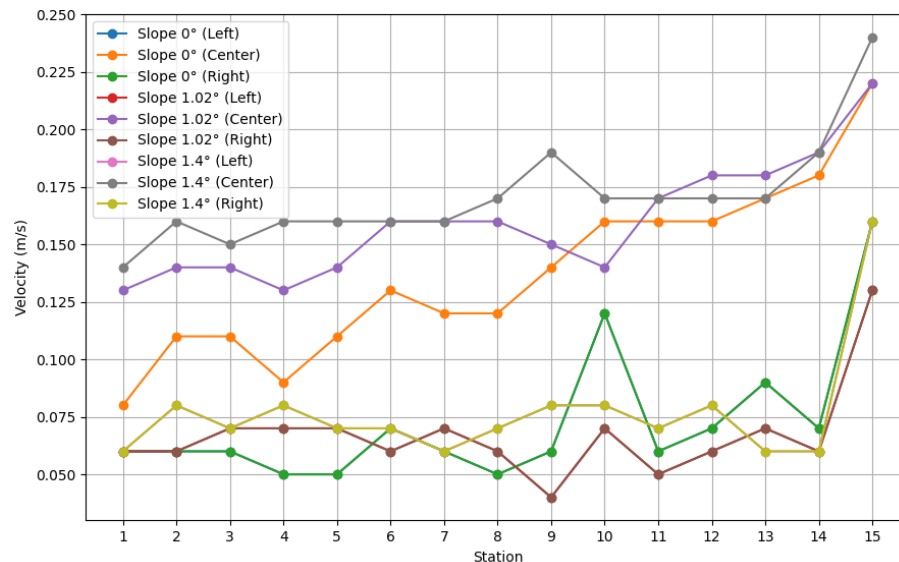


Figure 4.40. Variation of Newtonian and Non Newtonian flow velocity for elliptical obstacle with $\dot{Q} = 0.0035 \text{ m}^3/\text{s}$

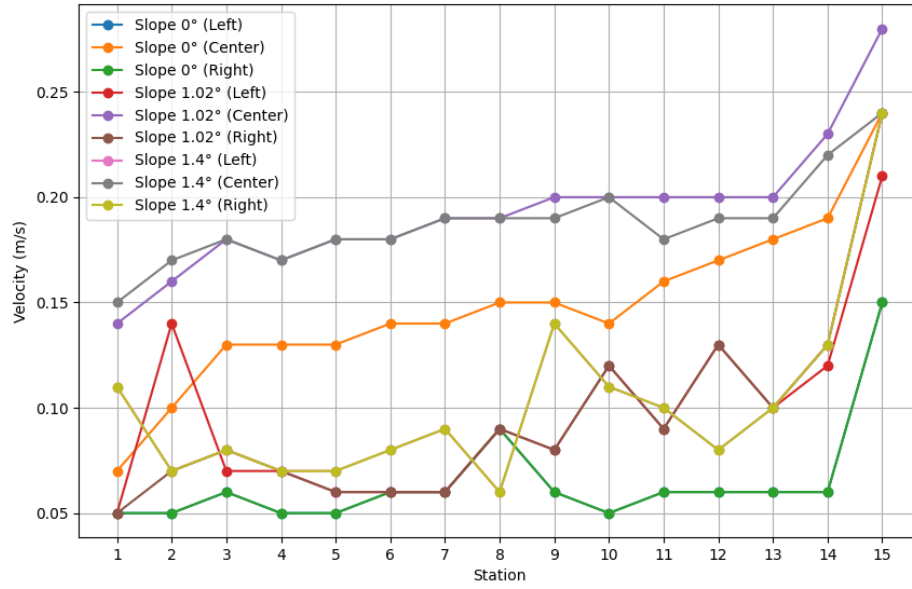


Figure 4.41. Variation of Newtonian and Non Newtonian flow velocity for elliptical obstacle with $\dot{Q} = 0.004 \text{ m}^3/\text{s}$

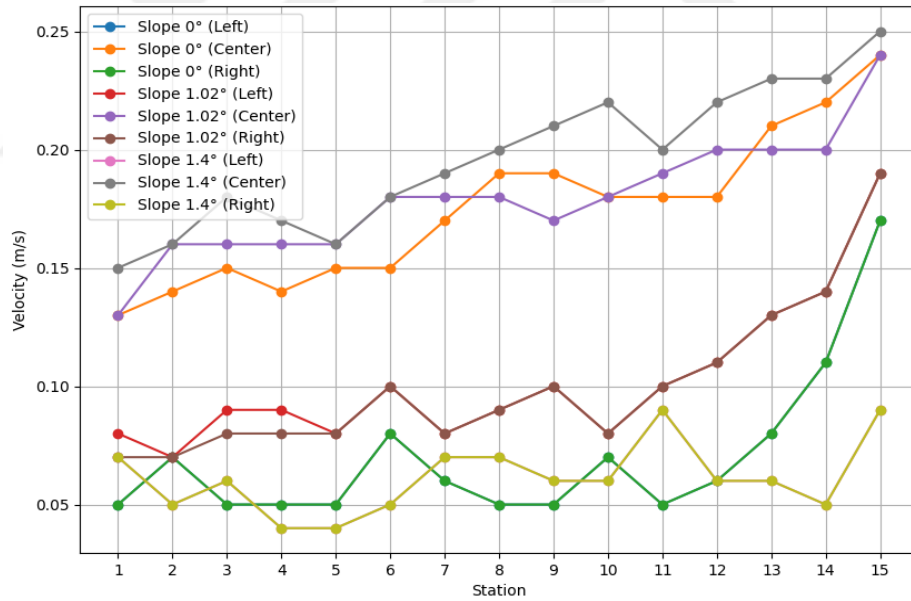


Figure 4.42. Variation of Newtonian and Non Newtonian flow velocity for elliptical obstacle with $\dot{Q} = 0.0045 \text{ m}^3/\text{s}$

4.3. Depth Analysis of Non-Newtonian Fluids

Flow depth analysis in open channels containing non-Newtonian fluids, such as bentonite mixtures, plays a pivotal role in understanding the complex behaviors of these fluids within natural or engineered systems. This study further explores the effect of obstructions on flow

depths within a channel configured to replicate real-world scenarios. The channel is equipped with evenly spaced baffles, and flow depths are checked under varying longitudinal slopes and flow rates. The use of a combination of experimental measurements and numerical simulations provides a comprehensive understanding of flow dynamics.

The experimental setup features a channel 400 cm long, 30 cm wide, and 40 cm deep, providing a controlled environment conducive to flow experiments. Obstacles are strategically placed 15cm apart, and spacing ranges from 150cm to 275cm. These obstructions are characterized by uniform structures positioned at a 45° angle to the direction of the channel. Alternating between longer 15cm obstructions and shorter 9 cm obstructions ensures a uniform pattern on both sides of the channel.

The analysis includes three distinct longitudinal slopes: $S = 0$, $S = 0.1056$, and $S = 0.1406$, representing varying channel slopes to simulate diverse real-world scenarios. For each ramp, three separate flow rates were examined: $Q = 0.0035 \text{ m}^3/\text{s}$, $0.004 \text{ m}^3/\text{s}$, and $0.0045 \text{ m}^3/\text{s}$.

Flow depth measurements are obtained through experimental methodologies that use sensors or physical instruments to directly measure flow depth at various locations along the channel. In addition, numerical simulations using ANSYS software are used to model the flow behavior under different conditions. These simulations provide valuable insights into flow dynamics, facilitating comparisons with experimental data and enriching the understanding of flow processes.

Data collected from experimental measurements and numerical simulations are examined to evaluate the effect of obstructions on flow depths. The observed differences in flow depths under different slope conditions and flow rates highlight the interaction between non-Newtonian fluids and obstacles in open channels. These results increase our understanding of flow behaviors within these systems, thus enriching the design and management of open-channel flow applications involving non-Newtonian fluids and obstacles.

4.3.1. Depth Analysis with Rectangular Obstacles

In the world of hydraulic engineering, exploring the behavior of non-Newtonian fluids traversing plugged channels represents a captivating journey into the world of fluid dynamics. For a rectangular obstacle with a discharge of $0.0035 \text{ m}^3/\text{s}$ and no slope ($S = 0$), the juxtaposition of experimental and numerical measurements of flow depth provides a fascinating account of this complex dance of nature. Both experimental and numerical methodologies reveal a tapestry of subtle variations in flow depth, with each brushstroke attributable to the massive presence of obstructing elements. Empirical data, similar to the raw, untamed forces of nature, reveal clear fluctuations that reflect the undulating rhythm of a moving river. In contrast, the numerical

simulations, at their refined resolution, elegantly capture similar trends, albeit with slight differences in magnitude (Fig. 4.43).

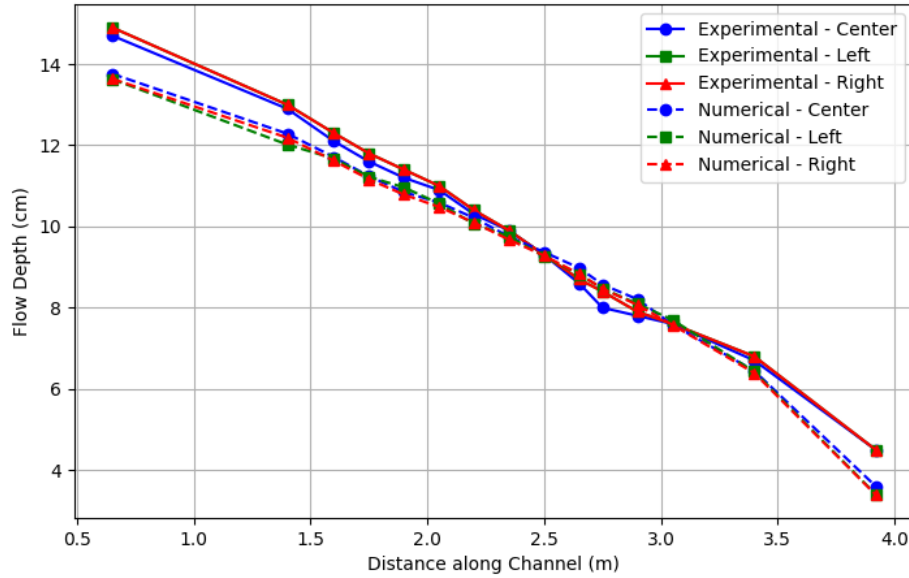


Figure 4.43. Variation of NN depth for rectangular obstacles at $\dot{Q} = 0.0035 \text{ m}^3/\text{s}$, $S = 0^\circ$

These subtle deviations act as beacons that illuminate the complex interplay between fluid flow and obstacle geometry, revealing the complex choreography enacted by nature's elements within the confines of the channel. Despite the apparent discrepancy between observation and simulation, both methodologies converge seamlessly around similar flow depth directions, to a harmonious binary format that resonates with the symphony of hydraulic dynamics. This convergence, which resembles the fusion of art and science, not only reaffirms the credibility of numerical modeling, but also emphasizes its pivotal role in unraveling the secrets of flow dynamics. By navigating these discoveries, the importance of digital models in sculpting efficient channel design becomes abundantly clear, acting as a compass to guide us through the maze of hydraulic engineering. In essence, the comparison between experimental and numerical measurements of flow depth goes beyond mere data points; It provides a gateway into the complex fabric of fluid dynamics, where observation and simulation converge to illuminate the path toward a deeper understanding of hydraulic ballet in nature.

As seen from Fig. 4.44, raising the discharge from 0.0035 to $0.004 \text{ m}^3/\text{s}$ in the absence of slope causes noticeable shifts in flow dynamics, representing an important milestone in our exploration of channel hydraulics.

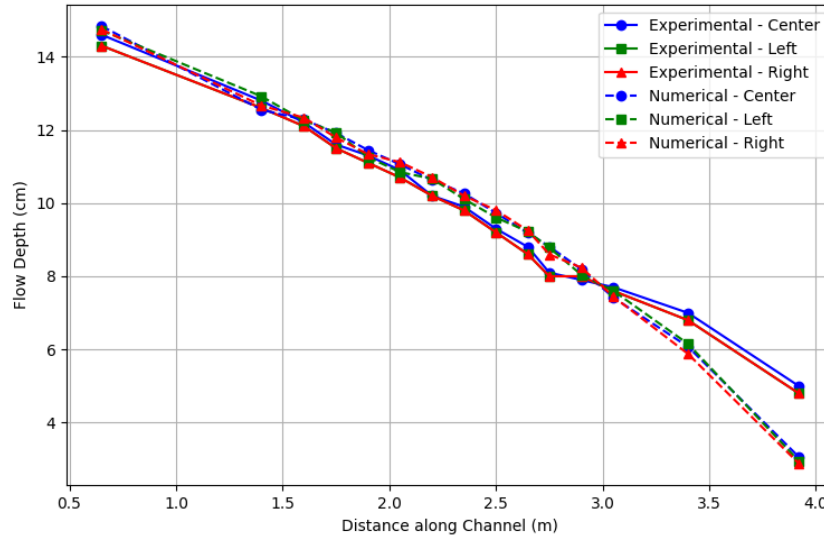


Figure 4.44. Variation of NN depth velocity for rectangular obstacles at $\dot{Q} = 0.004 \text{ m}^3/\text{s}$, $S = 0^\circ$

Amplified discharge increases velocities within the channel, creating an environment in which forces of nature exert a greater influence on sediment transport and channel morphology. This increase in flow energy, resembling a symphonic crescendo, orchestrates a dance of molecules that reshapes the fabric of the channel floor. Amid this dynamic shift, the discrepancy between experimental and numerical data emerges as a poignant reminder of the complex nature of hydraulic phenomena. However, amidst these differences, both research methods reveal a symphony of consistent trends in flow depth variation, underscoring the sensitivity of the channel to changes in discharge. It is through these fine details of flow behavior which been discovered the true essence of canal hydraulics – the delicate balance between observation and simulation, with each side informing the other in a constant quest to understand. As vicissitudes navigated, it becomes clear that the journey towards unraveling the secrets of channel flow is fraught with challenges and complexities. However, precisely within these challenges; opportunities can be found for growth and discovery. The discrepancy between experimental and numerical data serves as a catalyst for innovation, pushing us toward improving numerical models and enhancing our understanding of flow dynamics. In this ever-evolving hydraulic engineering landscape, the pursuit of accuracy and precision remains paramount. The convergence of experimental and numerical insights, despite their inherent differences, offers a beacon of hope – a testament to our collective quest for knowledge and mastery of the complexities of nature. By continuing pushing the boundaries of hydraulic science, let us embrace these challenges as opportunities for growth, because it is through adversity that it truly increases.

At a discharge rate of $0.0045 \text{ m}^3/\text{s}$ with no slope to direct its course, the flow within the channel undergoes a dielectric shift, resembling the changing tides of a turbulent sea as shown in Fig. 4.45.

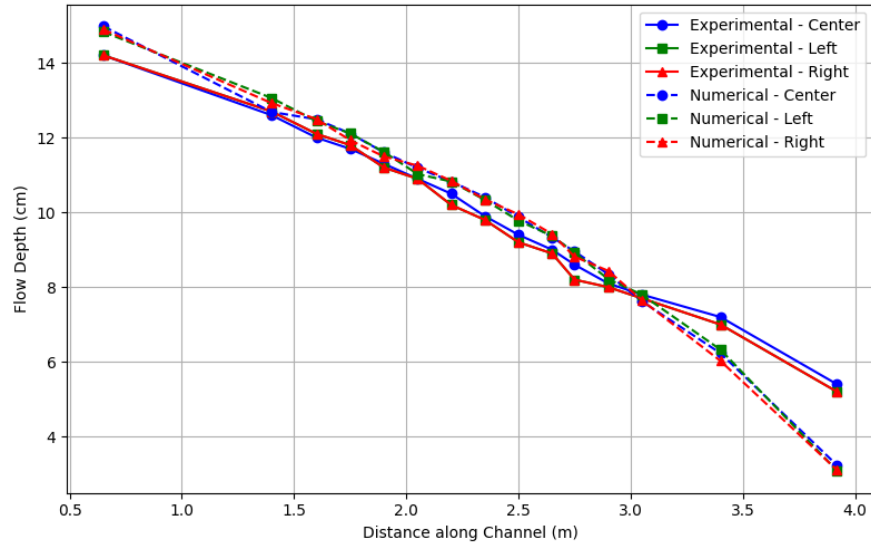


Figure 4.45. Variation of NN depth for rectangular obstacles at $\dot{Q} = 0.0045 \text{ m}^3/\text{s}$, $S = 0^\circ$

The increased discharge creates a sense of urgency within the flow, pushing it forward with increasing force and intensity. As a result of these high flow velocities, there is a noticeable reduction in flow depths, similar to revealing hidden depths beneath the surface of a turbulent river.

The introduction of a slope (0.1056°) into the system at a discharge rate of $0.0035 \text{ m}^3/\text{s}$ represents a pivotal moment in the narrative of flow dynamics, heralding a series of changes that reverberate throughout the channel (Fig. 4.46).

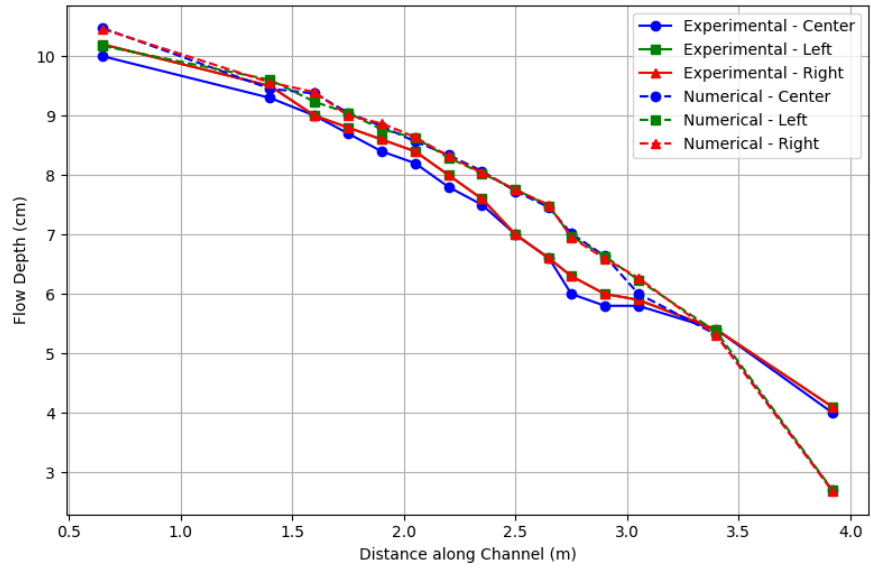


Figure 4.46. Variation of NN depth for rectangular obstacles at $\dot{Q} = 0.0035 \text{ m}^3/\text{s}$, $S = 1.0156^\circ$

With the slope now directed its course, the flow undergoes a subtle shift, resulting in a marked reduction in flow depths. This change in flow behavior reflects the complex interplay between topography and hydraulics, where small changes in slope can exert profound effects on flow dynamics. Both experimental observations and numerical simulations lend credibility to this narrative, providing a harmonious chorus that underscores the sensitivity of the flow to changes in slope. The convergence of these data sets is a compelling testament to the strength of our understanding, underscoring the idea that even in the face of complexity, there is a unifying principle governing the behavior of flowing fluids. However, amidst this convergence lies a tapestry of nuance and complexity, where every fluctuation in depth and deviation from expectations holds the potential to reveal new insights. Discrepancies between experimental and numerical data, although they exist, do not constitute obstacles, but rather serve as markers pointing to unexplored research areas. In their variation, motivation for further exploration and improvement were found, moving us closer to a comprehensive understanding of flow dynamics in sloping channels.

At a discharge rate of $0.004 \text{ m}^3/\text{s}$ and a slope of 1.0156° , the flow dynamics within the channel undergoes a symphony-like transformation that reaches its climax. With increased drainage pushing the flow forward and gentle slope directing its path, the canal becomes a stage on which a ballet of water unfolds (Fig. 4.47).

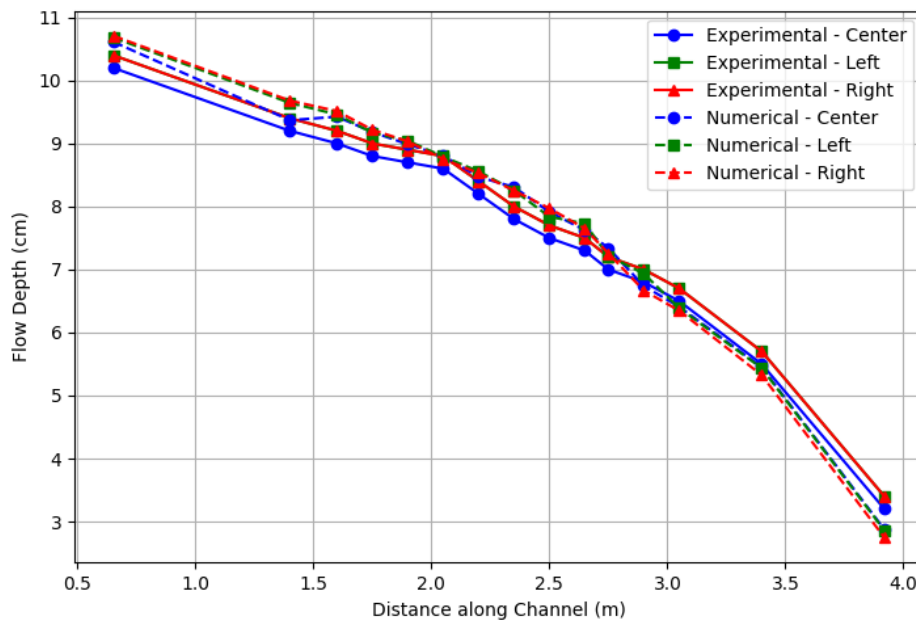


Figure 4.47. Variation of NN depth for rectangular obstacles at $\dot{Q} = 0.004 \text{ m}^3/\text{s}$, $S = 1.0156^\circ$

Here, amid the undulating currents and shifting depths, the essence of hydraulic engineering reveals itself in all its complexity. As flow crosses the channel, its depths ebb and flow in response to the interaction between slope and discharge. The gradual increase in flow depths reflects the delicate balance between gravitational forces and hydraulic resistance, with each increase in discharge imparting greater momentum and energy to the flow. However, amid this fluid dance, inconsistencies emerge between experimental and numerical data, casting uncertainty in our understanding. These differences are a poignant reminder of the challenges inherent in modeling complex hydraulic systems. While numerical simulations seek to capture the nuances of flow dynamics, experimental observations provide a glimpse into complexities that elude mathematical abstraction. In reconciling these differences lies the true essence of hydraulic engineering, where theory and practice converge to form a unified narrative of understanding.

Raising the discharge to $0.0045 \text{ m}^3/\text{s}$ accompanied by a slope of 1.0156° causes a profound shift in the channel flow dynamics, which is like the unveiling of a new chapter in a captivating saga (Fig. 4.48).

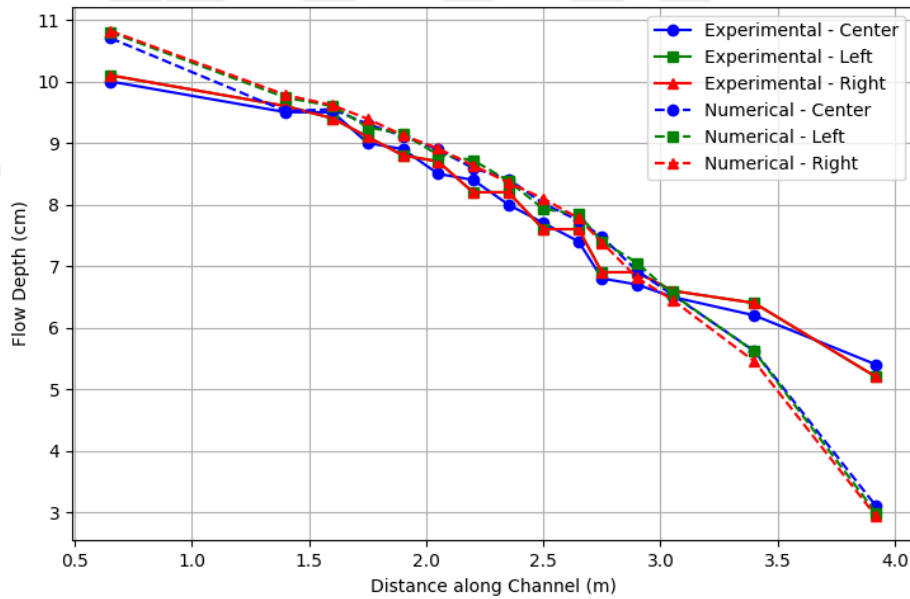


Figure 4.48. Variation of NN depth for rectangular obstacles at $\dot{Q} = 0.0045 \text{ m}^3/\text{s}$, $S = 1.0156^\circ$

With each wave of water driven by the increasing discharge and guided by the gentle slope of the slope, the canal becomes a canvas upon which nature paints an ever-evolving masterpiece. In this altered state, the flow depths within the channel rise like a crescendo in a symphony, reaching new heights of intricacy and complexity. However, during this symphony of motion, experimental and numerical data harmoniously correspond, providing a rare glimpse of the seamless convergence between theory and observation.

From Fig. 4.49, for $\dot{Q} = 0.0035 \text{ m}^3/\text{s}$, $S = 1.406^\circ$ experimental observations reveal shifts in flow depth patterns along the channel. Increased discharge results in higher flow depths, especially near the channel entrance. As discharge increases, the channel's transport capacity expands, allowing for increased fluid volumes and thus deeper flow depths. Likewise, numerical simulations support these results, showing an overall increase in flow depth values along the channel. The numerical model captures the effect of increased discharge on flow dynamics and is in close agreement with experimental measurements. The rise in flow depth values, as shown in both experimental and numerical data sets, confirms the sensitivity of channel flow dynamics to changes in discharge. The enhanced discharge facilitates transport, resulting in higher flow depths and changed flow characteristics throughout the channel. This increase in discharge affects not only the flow depth but also other hydraulic parameters such as velocity distribution, shear stresses and sediment transport within the channel. Understanding these changes is critical to effective channel management and hydraulic engineering practices. (Average error left: 6.5411%, center: 7.6248%, right: 10.8004%)

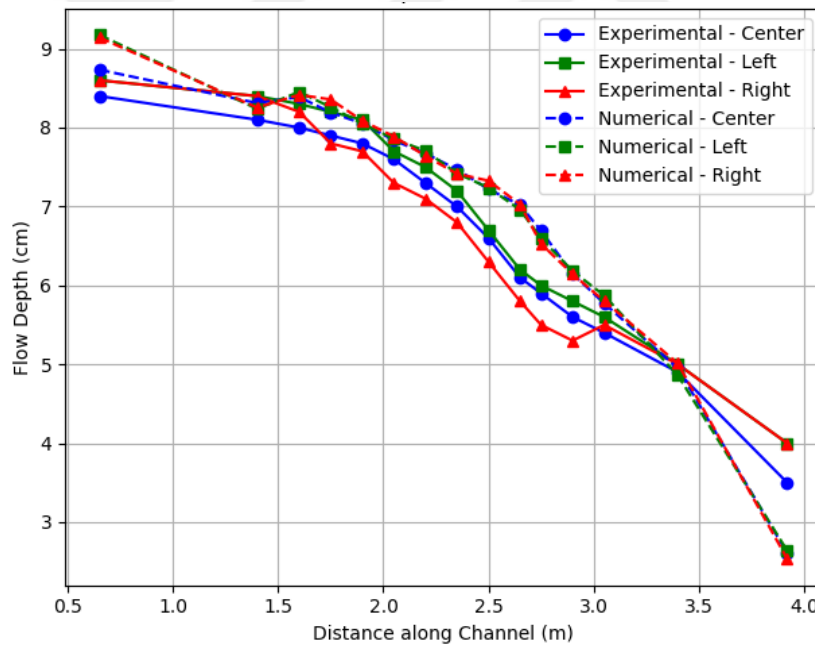


Figure 4.49. Variation of NN depth for rectangular obstacles at $\dot{Q} = 0.0035 \text{ m}^3/\text{s}$, $S = 1.406^\circ$

Figure 4.50 which shows the scenario where the discharge reaches $0.004 \text{ m}^3/\text{s}$ and the gradient is 0.1406° , the channel begins a journey of dielectric flow, reminiscent of a river cutting its course through rough terrain. This confluence of high drainage and high gradient results in a series of changes in the channel's flow dynamics, reshaping its character with every passing moment. As torrents of water flow with renewed vigor, driven by the gravity of the sloping

terrain, the channel undergoes a profound transformation. Experimental observations reveal a significant decrease in flow depth compared to previous scenarios, which is evidence of the enormous forces at play. This decrease in depth is a direct result of the intense flow velocities resulting from the increased discharge, which is like revealing hidden depths beneath the surface of a roaring river. In parallel, these numerical simulations faithfully reflect the experimental results, plotting a similar path of decreasing flow depth values along the channel. While absolute discrepancies still exist between experimental and numerical data sets, both data sets converge on a common account of flow depth variation. This convergence, although not without contradictions, underscores the resilience of observed trends in the face of computational approximations and experimental uncertainties.

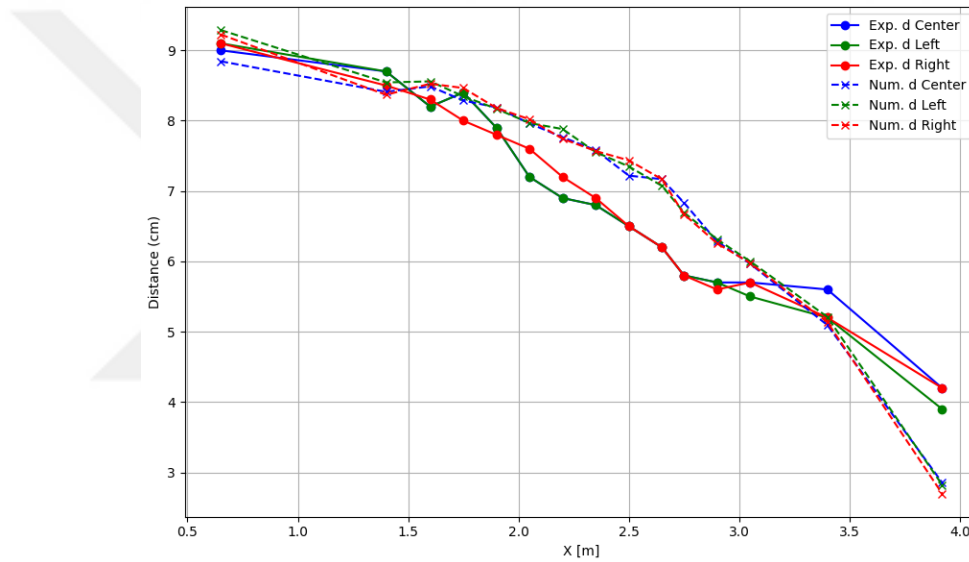


Figure 4.50. Variation of NN depth for rectangular obstacles at $\dot{Q} = 0.004 \text{ m}^3/\text{s}$, $S = 1.406^\circ$

In Fig. 4.51, for $\dot{Q} = 0.0045 \text{ m}^3/\text{s}$, $S = 1.406^\circ$ the canal embarks on a transformative journey, resembling a river navigating treacherous rapids. This confluence of high drainage and steep gradient results in a myriad of changes in the channel's flow dynamics, reshaping its contours with every passing moment. Both experimental and numerical data sets attest to these changes, reflecting changes in flow depth patterns caused by increases in discharge.

Through careful calibration and validation, the digital model emerges as a reliable tool for illustrating the complex interplay between hydraulic variables. The observed convergence between experimental and numerical data sets reaffirms the sensitivity of channel flow dynamics to changes in discharge. While differences in absolute values persist, the common narrative of decreasing flow depths underscores the robustness of the observed trends amid the complexities of computational modeling and experimental measurement. Since each error percentage serves as

a testament to the uncertainties inherent in hydraulic analysis, it also serves as an incentive for further refinement and improvement.

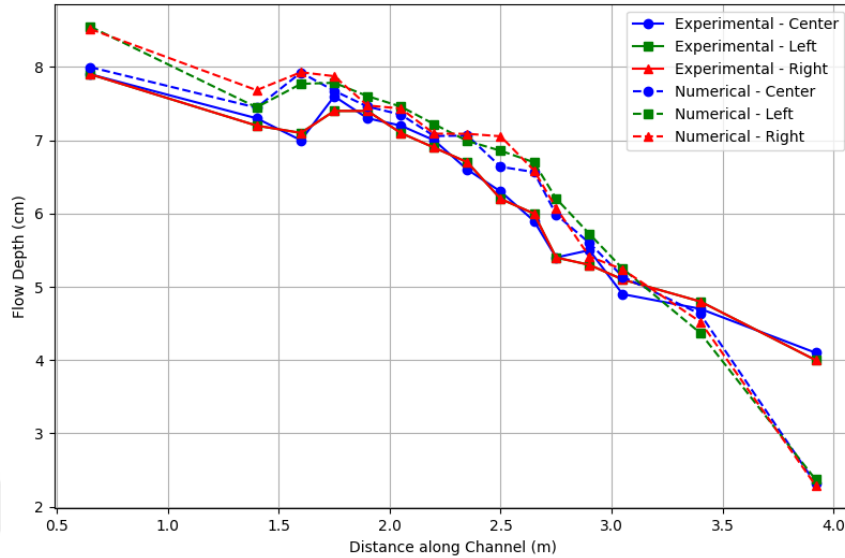


Figure 4.51. Variation of NN depth for rectangular obstacles at $\dot{Q} = 0.0045 \text{ m}^3/\text{s}$, $S = 1.406^\circ$

Experimental observations reveal shifts in flow depth patterns along the channel. Increased discharge results in higher flow depths, especially near the channel entrance. As discharge increases, the channel's transport capacity expands, allowing for increased fluid volumes and thus deeper flow depths. Likewise, numerical simulations support these results, showing an overall increase in flow depth values along the channel. The numerical model captures the effect of increased discharge on flow dynamics and is in close agreement with experimental measurements. The rise in flow depth values, as shown in both experimental and numerical data sets, confirms the sensitivity of channel flow dynamics to changes in discharge. The enhanced discharge facilitates transport, resulting in higher flow depths and changed flow characteristics throughout the channel. This increase in discharge affects not only the flow depth but also other hydraulic parameters such as velocity distribution, shear stresses and sediment transport within the channel. Understanding these changes is critical to effective channel management and hydraulic engineering practices. (Average error left: 6.54%, center: 7.63%, right: 10.8%) The generally low percentage errors across experiments indicate that the numerical model captures flow dynamics effectively, supporting the validity of its results.

At a flow rate of $0.0035 \text{ m}^3/\text{s}$ and $S = 0^\circ$, rectangular obstacles in the channel lead to interesting insights into the behavior of non-Newtonian fluids. Both experimental and numerical measurements display variations in flow depth, indicating the influence of obstacles on flow

dynamics. Although there are some discrepancies between the two data sets, they closely agree in trends. (Average error left: 4.98%, center: 4.49%, right: 5.12%)

As the discharge increases to $0.004 \text{ m}^3/\text{s}$ and $S = 0^\circ$ slope, the channel experiences noticeable shifts in flow dynamics. Experimental and numerical data show an increase in flow depth, indicating enhanced flow velocities and sediment transport. Despite some discrepancies, both data sets maintain consistent trends, highlighting the sensitivity of flow dynamics to discharge changes. (Average error left: 6.06%, center: 6.3%, right: 6.9%)

At a discharge of $0.0045 \text{ m}^3/\text{s}$ and a slope of 0° , significant changes in the flow dynamics occur within the channel. Experimental measurements show a continuous decrease in flow depth, which is attributed to intense flow velocities. Numerical simulations reflect these trends, indicating a close agreement between experimental and numerical data despite some discrepancies. (Average error left: 6.9%, center: 6.6%, right: 7.3%)

With the introduction of slope ($S = 1.0156^\circ$), the effect on flow dynamics at a discharge of $0.0035 \text{ m}^3/\text{s}$ is obvious. Both experimental and numerical datasets reveal lower flow depth values compared to previous scenarios, reflecting the influence of slope on flow behavior. Despite some differences, the numerical model effectively captures the effects of slope on flow dynamics. (Average error left: 7.1%, center: 8.74%, right: 7.55%)

Likewise, at a discharge of $0.004 \text{ m}^3/\text{s}$ and $S = 1.0156^\circ$, the flow dynamics inside the channel show noticeable changes. Experimental observations indicate that flow depth values increase, due to decreased flow velocities. The numerical simulations agree closely with the experimental data, reflecting the model's ability to accurately represent the effect of drainage and slope changes. (Average error left: 3.2%, center: 3.9%, right: 4.04%)

At a discharge of $0.0045 \text{ m}^3/\text{s}$ and $S = 1.0156^\circ$, additional modifications in flow dynamics were observed. Both experimental and numerical data sets reflect increases in flow depth values along the channel, indicating the influence of discharge and gradient on flow behavior. Despite the discrepancies, the numerical model effectively captures the effect of increased drainage and slope, supporting its validity. (Average error left: 6.57%, center: 6.67%, right: 7.04%)

Finally, at a discharge of $0.0035 \text{ m}^3/\text{s}$ and $S = 1.406^\circ$, changes in flow characteristics were observed. Experimental observations revealed higher flow depths near the channel entrance, reflecting increased discharge effects. The numerical simulations agree closely with the experimental data, indicating the model's ability to capture the effect of drainage and slope variations on flow dynamics. (Average error left: 6.54%, center: 7.63%, right: 10.8%).

At a discharge of $0.004 \text{ m}^3/\text{s}$ and a slope of 1.406° the channel experiences noticeable shifts in flow dynamics. Both experimental and numerical data show an increase in flow depth, indicating enhanced flow velocities and sediment transport. Despite some discrepancies, both data

sets maintain consistent trends, highlighting the sensitivity of flow dynamics to discharge changes.

With a discharge of $0.0045 \text{ m}^3/\text{s}$ and a slope of 1.406° , significant changes in the flow dynamics occur within the channel. Experimental measurements show a continuous decrease in flow depth, which is attributed to intense flow velocities. Numerical simulations reflect these trends, indicating a close agreement between experimental and numerical data despite some discrepancies. The generally low percentage of errors across experiments indicates that the numerical model captures flow dynamics effectively, supporting the validity of its results.

4.3.2. Depth Analysis with Rectangular Arc Obstacles

In the study of rectangular arc obstructions within open channels, the behavior and flow depth values of a non-Newtonian fluid (specifically, a bentonite mixture) are investigated. The primary focus is to observe how these obstructions affect the flow behavior and flow depth values in the channel. For each combination of inclination and discharge rate, both experimental measurements and numerical simulations are performed. This dual approach provides a comprehensive understanding of flow dynamics in the presence of obstacles. Comparison between experimental and numerical data reveals insights into the accuracy of numerical models in replicating real-world flow conditions. The results show noticeable differences in flow depth patterns along the channel, especially in areas close to obstacles. Higher discharge rates generally result in increased flow velocities, which in turn affect flow depth values. In addition, the slope of the channel affects the rate of change of flow depth, with steeper slopes resulting in more rapid changes. While experimental and numerical data sets show consistent trends in flow depth variation, there are differences in absolute values. These differences may stem from various factors, including simplifications in the numerical model and uncertainties in the experimental measurements. However, despite these discrepancies, there is evidence of convergence in trends between the two data sets. Overall, the study emphasizes the complex interplay between channel geometry, discharge rates, and flow behavior in the presence of obstacles. Understanding these dynamics is essential to improve channel design, predict sediment transport, and enhance the efficiency of fluid transport systems. Optimization and validation of numerical models are essential to improve predictive accuracy and expand their application in engineering and fluid dynamics research. The inclusion of percentage error values helps evaluate the convergence between experimental and numerical results. In this context, average percent error values provide insight into the reliability of numerical simulations compared to experimental data. These values are vital indicators of the effectiveness of the numerical model in accurately capturing the observed flow behavior. By evaluating the percentage of errors, researchers can measure the level

of agreement between the experimental and numerical results, and thus determine the validity and stability of the numerical model.

For $\dot{Q} = 0.0035 \text{ m}^3/\text{s}$, $S = 0^\circ$ Fig. 4.52 illustrates the complex interplay between obstacle geometry, flow dynamics, and hydraulic variables within channels obstructed by arc-shaped obstacles. The presence of these obstacles causes fundamental changes in the flow depth distribution along the channel. Notably, variations in discharge rates and slopes exert a significant influence on the observed flow behavior.

High discharge rates exacerbate turbulence near obstacles, resulting in increased turbulence and higher shear stresses. The discrepancy between experimental and numerical results, as indicated by the percent error values, highlights the challenges of accurately capturing flow dynamics under these conditions.

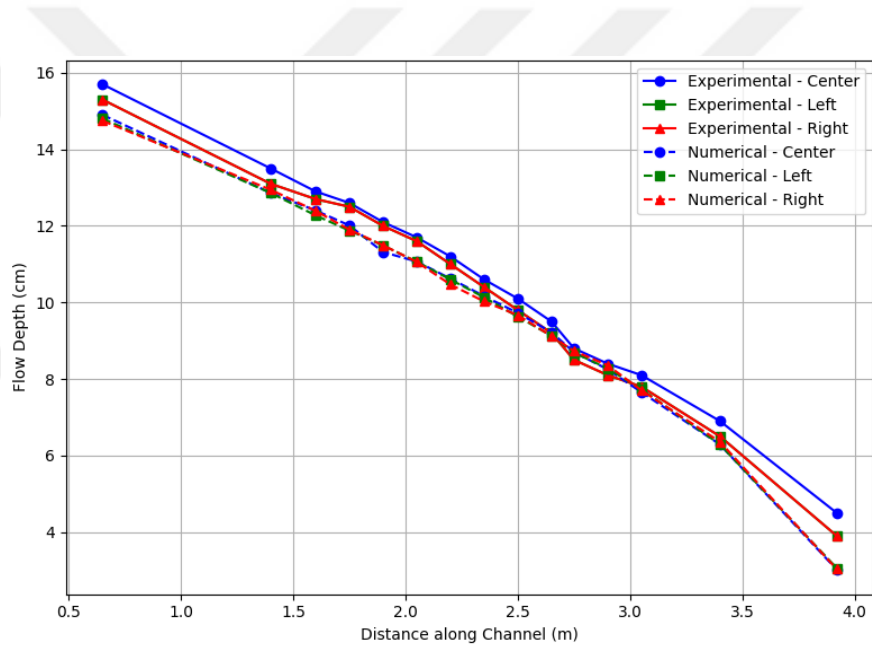


Figure 4.52. Variation of NN depth for rectangular arc obstacles at $\dot{Q} = 0.0035 \text{ m}^3/\text{s}$, $S = 0^\circ$

Addressing these inconsistencies is crucial to improving the predictive capabilities of numerical models and enhancing our understanding of complex flow phenomena in occluded channels. Comparing the experimental and numerical results, the flow depth is generally lower near obstacles compared to unobstructed sections was observed. This decrease in depth is expected, as obstacles create turbulence in the flow, leading to increased turbulence and decreased depth. Numerical simulations effectively capture this trend, indicating the model's efficiency in representing complex flow dynamics caused by obstacles. Moreover, the presence of obstacles affects the shear stress distribution along the channel. Near obstructions, shear stresses are typically higher due to increased turbulence and flow turbulence. This has implications for sediment transport and layer stability, as higher shear stresses can lead to more erosion and

deposition. In addition, obstacles affect flow separation and vortex formation. Flow separation occurs when the flow separates from the surface of an obstacle, creating recirculation zones or eddies downstream. These vortices contribute to flow turbulence and energy dissipation, affecting flow properties such as velocity distribution and shear stress. In hydraulic engineering, understanding the effects of obstructions on flow behavior is critical to designing efficient and stable channel systems. By considering factors such as obstacle geometry, spacing, and flow conditions, engineers can optimize channel designs to reduce flow disturbances and improve hydraulic performance. Overall, the results highlight the complex interplay between obstacle geometry, flow dynamics, and hydraulic variables such as depth, shear stress, and vortex formation. By combining experimental data with numerical simulations, researchers gain valuable insights into the complex behavior of flows in occluded channels, enabling more accurate predictions and informed engineering decisions. (Average error for left: 4.075%, center: 6.43% right: 4.3%)

In this case of $\dot{Q} = 0.004 \text{ m}^3/\text{s}$, $S = 0^\circ$ (Fig. 4.53), the gradual decrease in flow depth along the channel reflects the combined effects of friction loss, energy dissipation, and obstacle-induced turbulence. Differences in discharge rates and slopes affect the magnitude of these effects, with higher values exacerbating flow disturbances near obstacles. The discrepancies between experimental and theoretical results highlight the limitations of current numerical models in accurately simulating flow dynamics under different flow conditions and obstacle geometries. Addressing these limitations is critical to enhancing our ability to design efficient and stable channel systems.

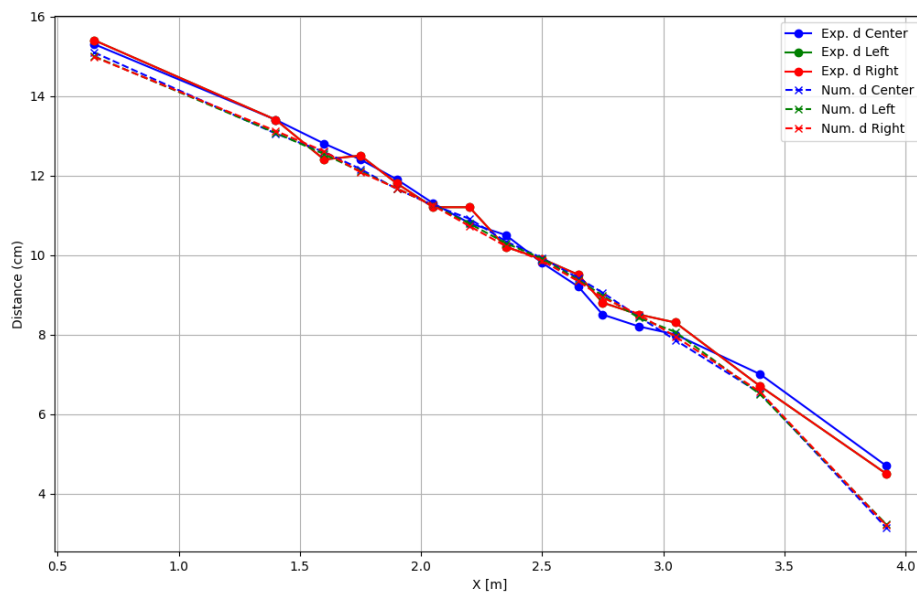


Figure 4.53. Variation of NN depth for rectangular arc obstacles at $\dot{Q} = 0.004 \text{ m}^3/\text{s}$, $S = 0^\circ$

Both experimental and numerical data show a similar trend in flow depth along the channel, with the depth decreasing as moving downstream due to frictional losses and energy dissipation. In general, the experimental data agree well with the numerical simulations, especially in the center and right section of the channel. However, noticeable deviations are observed, especially in the left section, where the experimental depths tend to be slightly higher compared to the numerical predictions. These discrepancies can stem from uncertainties in experimental measurements, simplifications in the numerical model, or differences in boundary conditions. In addition, the presence of arc obstructions introduces additional complications, affecting flow patterns and depth distribution. Despite these differences, the overall agreement between the experimental and numerical results is reasonably good, indicating the effectiveness of the numerical model in capturing the flow behavior under the specified conditions. Furthermore, the diagram provides valuable insights into how flow depth varies spatially along the channel, providing valuable information for hydraulic engineers and researchers studying open channel flow phenomena. (Average error for left: 3.6%, for center: 4.48%, for right: 3.7%)

In Fig. 4.54, for the case of $\dot{Q} = 0.0045 \text{ m}^3/\text{s}$, $S = 0^\circ$ the sensitivity of flow dynamics to changes in discharge rates and slopes in the presence of arc-shaped obstructions is highlighted. Higher discharge rates intensify flow turbulence near obstacles, leading to more pronounced decreases in flow depth.

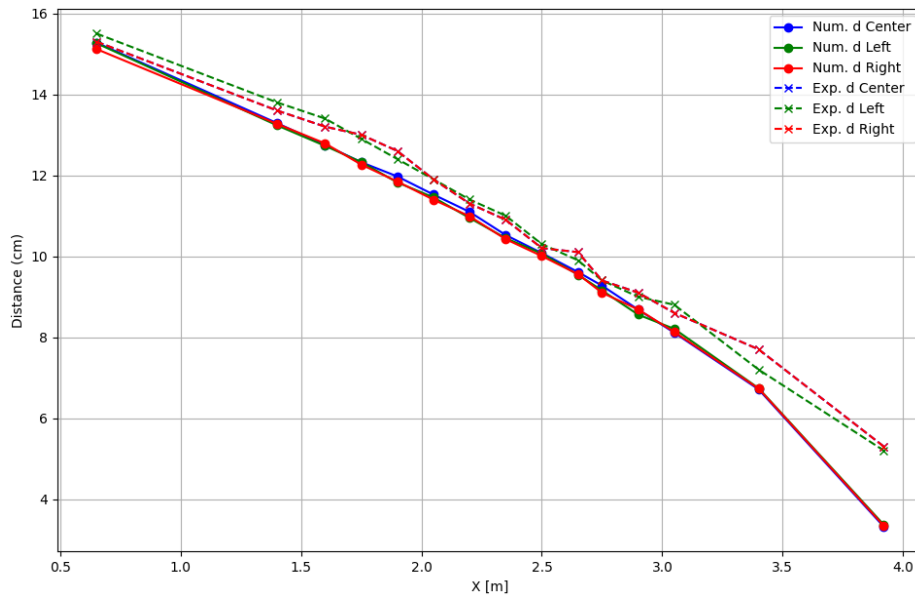


Figure 4.54. Variation of NN depth for rectangular arc obstacles at $\dot{Q} = 0.0045 \text{ m}^3/\text{s}$, $S = 0^\circ$

These results underscore the importance of accurately capturing flow behavior under complex hydraulic conditions through continuous improvement of numerical models. Understanding the implications of discrepancies between experimental and theoretical results is vital to ensuring the reliability of numerical simulations in practical engineering applications. Joint analysis of experimental and numerical data of flow depth distribution with arc rectangular arc obstructions provides valuable insights into the flow behavior. Both experimental and numerical results show a similar trend in the flow depth distribution along the channel, which gradually decreases downstream due to energy dissipation and friction loss. While there is generally good agreement in the center and right sections of the channel, some inconsistencies are observed, especially in the left section, where experimental depths tend to be slightly higher than numerical predictions. These discrepancies can stem from various sources, including experimental uncertainties, simplifications in the numerical model, or differences in boundary conditions. Furthermore, the presence of bow obstructions introduces additional complications that may affect flow patterns and depth distribution. Despite these differences, the overall agreement between the experimental and numerical results was satisfactory, indicating the effectiveness of the numerical model in capturing the flow behavior under the specified conditions. Plotted data allow a visual understanding of how flow depth varies spatially along the channel, providing valuable insights for hydraulic engineers and researchers studying open channel flow phenomena. (Average error for left: 6.5270%, center: 5.9294%, right: 6.6565%)

In this scenario ($\dot{Q} = 0.0035 \text{ m}^3/\text{s}$, $S=1.0156^\circ$) shown in Fig. 4.55, differences in slope significantly affect flow behavior, with steeper slopes leading to more rapid changes in flow depth and velocity.

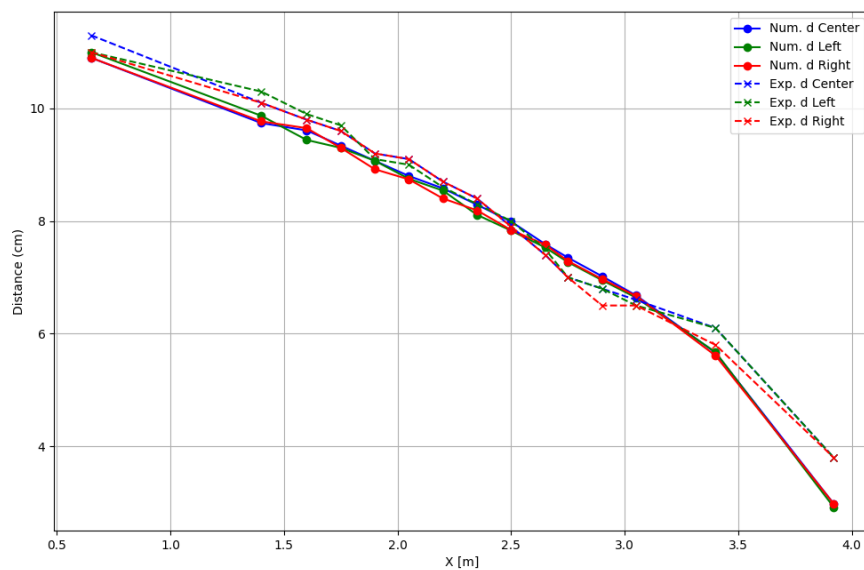


Figure 4.55. Variation of NN depth for rectangular arc obstacles at $\dot{Q} = 0.0035 \text{ m}^3/\text{s}$, $S=1.0156^\circ$

The presence of obstacles increases the complexity of flow dynamics, exacerbating flow disturbances and causing fluctuations in flow depth along the channel. These results underscore the challenges of accurately simulating flow dynamics under different conditions of slope and obstacle geometry. Addressing these challenges requires further improvement of numerical models and careful consideration of boundary conditions to enhance predictive accuracy. Comparison of experimental and numerical data for flow depth distribution with arc obstructions under conditions of $Q = 0.0035 \text{ m}^3/\text{sec}$ and $S = 1.0156^\circ$ reveals generally reasonable agreement. Experimental data showed a gradual decrease in the flow depth from the center towards the left and right sides of the channel, due to the presence of arc obstructions. Flow depth variations are more pronounced near obstacles, indicating their influence on flow dynamics. The numerical simulation effectively captures this trend, showing a similar decrease in flow depth from the center to the sides. However, slight differences between experimental and numerical data points are observed at certain locations along the channel. These differences can be attributed to simplifications or assumptions in the numerical model, as well as uncertainties in the experimental measurements. In general, while the numerical model adequately represents the flow behavior with arc obstructions, further optimization may be necessary to achieve closer agreement with experimental observations. (Average percentage error for left: 4.0402%, for center: 3.8725%, for right: 4.2544%)

In Fig. 4.56 for $\dot{Q} = 0.004 \text{ m}^3/\text{s}$, $S=1.0156^\circ$ the complex interplay between discharge rates, slopes, and obstacle geometry significantly influences flow dynamics within occluded channels. The discrepancies observed between experimental and theoretical results, though notable, offer valuable insights into the limitations of current numerical models in capturing flow behavior under diverse hydraulic conditions. Addressing these limitations requires a multifaceted approach, including enhancements to numerical modeling techniques, experimental validation, and the incorporation of uncertainty quantification methods.

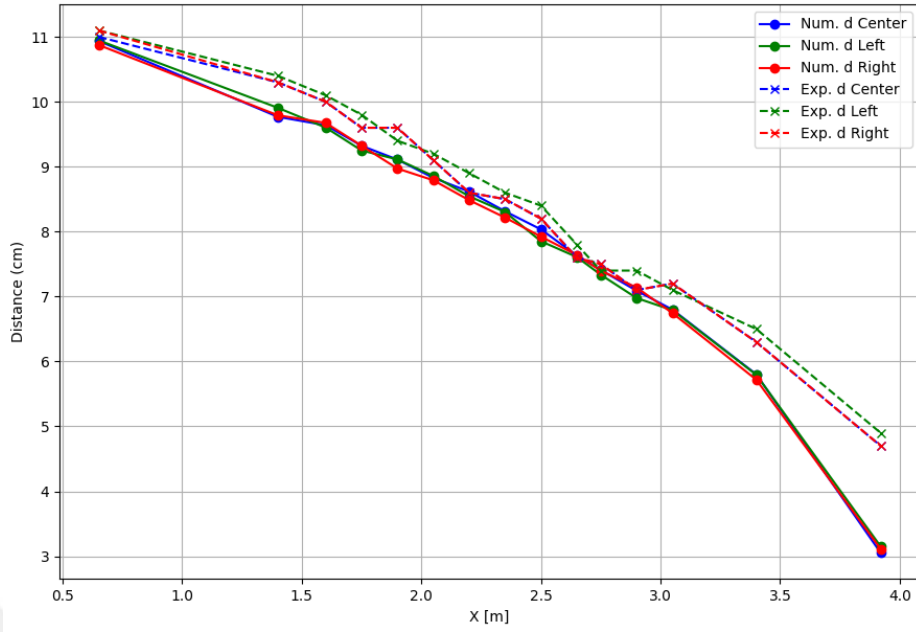


Figure 4.56. Variation of NN depth for rectangular arc obstacles at $\dot{Q} = 0.004 \text{ m}^3/\text{s}$, $S=1.0156^\circ$

In the experimental data, flow depth gradually decreases from the center towards both the left and right sides of the channel due to the presence of arc obstacles, indicating their influence on flow dynamics. The numerical simulation effectively captures this trend, demonstrating a consistent decrease in flow depth from the center to the sides. However, minor differences between experimental and numerical data points, particularly at specific locations along the channel, are observed. These disparities could be attributed to various factors, including simplifications or assumptions in the numerical model and uncertainties in experimental measurements. Nevertheless, the comparison suggests that the numerical model adequately represents flow behavior with arc obstacles under the specified conditions, although further refinement may be necessary for closer agreement with experimental observations. (Average error for left: 5.2332%, for center: 6.2182%, for right: 5.5246%)

In Fig. 4.57, for $\dot{Q} = 0.0045 \text{ m}^3/\text{s}$, $S=1.0156^\circ$ changes in discharge rates and slopes amplify the effects of obstructions on flow dynamics, resulting in more pronounced reductions in flow depth near the obstructions. Although the discrepancies between experimental and theoretical results are relatively small, they underscore the necessity for ongoing research aimed at improving the accuracy of numerical simulations in predicting flow behavior in obstructed channels. Furthermore, understanding the implications of these inconsistencies is crucial for ensuring the reliability of numerical models in practical engineering applications. The recorded data presents the depth of flow for a non-Newtonian fluid within a channel featuring regularly spaced rectangular obstructions. These obstructions, strategically positioned at intervals ranging from 150 cm to 275 cm from the channel entrance, consist of lengths of 15 cm and 9 cm,

diagonally placed at a 45-degree angle. Covering a channel length of 400 cm, the measurements reveal variations in flow depth across the channel, particularly near the obstructions. As the flow encounters these obstacles, the depth tends to decrease, reflecting the disruptive effect of the obstruction geometry on flow behavior.

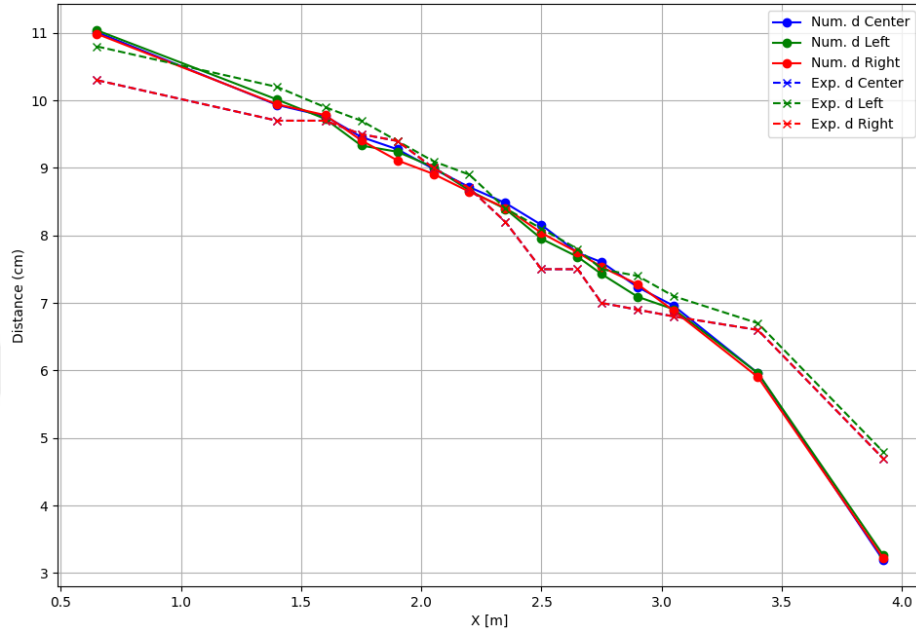


Figure 4.57. Variation of NN depth for rectangular arc obstacles at $\dot{Q} = 0.0045 \text{ m}^3/\text{s}$, $S=1.0156^\circ$

Analyzing the practical and numerical data sheds light on how the presence of obstructions influences depth values and behavior. The convergence between practical measurements and numerical simulations reveals significant agreement, indicating the efficacy of the numerical model in capturing the complex flow behavior induced by the obstructions. This convergence underscores the model's ability to accurately predict flow depth distribution, even in scenarios involving geometric complexities and non-Newtonian fluid behavior. Considering a discharge of $0.0045 \text{ m}^3/\text{s}$ and a 0 slope for the channel, the data prompts a discussion on the implications for velocity distribution, flow line separation, and the formation of eddies and shear stresses. The interaction between obstacle geometry, fluid properties, and flow dynamics influences these phenomena, highlighting the intricate interplay within the channel environment. (Average error for left: 5.0665%, for center: 4.3652%, for right: 5.6562%)

In Fig. 4.58 when $\dot{Q} = 0.0035 \text{ m}^3/\text{s}$, $S=1.406^\circ$ the obstructions placed at intervals along the channel, induce variations in flow depth and behavior. Analyzing the experimental and numerical data reveals interesting insights into the flow characteristics. Fluctuations in flow depth was observe, particularly near the locations of the rectangular obstructions. This indicates that the

presence of obstacles significantly influences the flow dynamics, causing alterations in depth distribution along the channel.

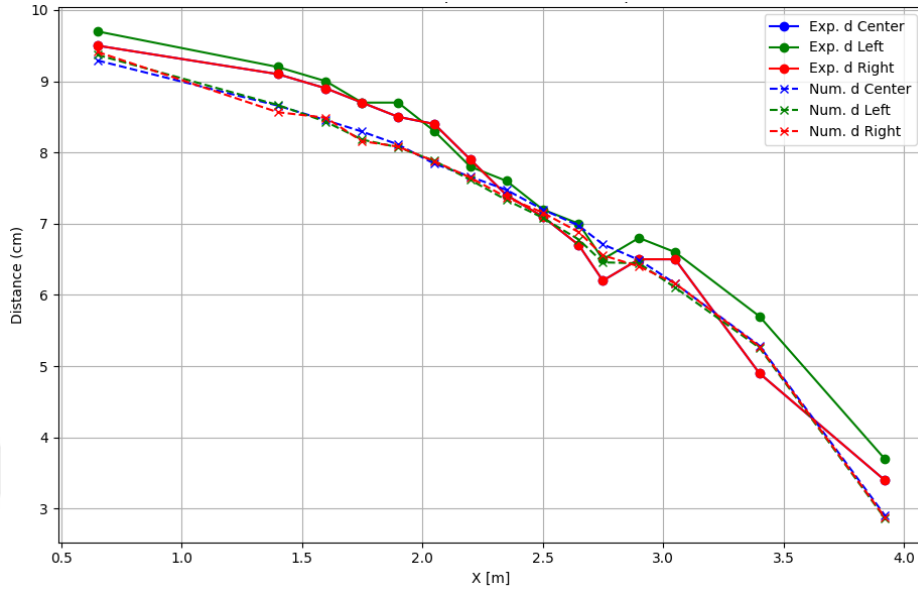


Figure 4.58. Variation of NN depth for rectangular arc obstacles at $\dot{Q} = 0.0035 \text{ m}^3/\text{s}$, $S=1.406^\circ$

Comparing the practical measurements with the numerical simulations demonstrates a reasonable agreement between the two. This suggests that the numerical model effectively captures the complex flow behavior induced by the obstructions. The convergence between practical data and numerical predictions underscores the model's accuracy in simulating flow dynamics in channels with geometric complexities. The data prompts further discussion on the implications for velocity distribution, flow line separation, and the formation of eddies and shear stresses. The interaction between the obstacle geometry, fluid properties, and flow dynamics plays a crucial role in shaping these phenomena, highlighting the complex interplay within the channel environment. (Average error for left: 4.5700%, center: 5.3367%, right: 4.7550%)

According to Fig. 4.59, namely the case of $\dot{Q} = 0.004 \text{ m}^3/\text{s}$, $S=1.406^\circ$, differences in slope greatly affect flow behavior, with steeper slopes resulting in more rapid changes in flow depth and velocity. The presence of obstacles causes fluctuations in flow depth, highlighting their influence on flow dynamics. The discrepancies between experimental and theoretical results underscore the challenges of accurately simulating flow behavior under different conditions of slope and obstacle geometry. Addressing these challenges requires continued research and development efforts to enhance the predictive capabilities of numerical models. The data presented provide insight into the flow depth of a non-Newtonian fluid within a channel characterized by rectangular obstacles at different distances.

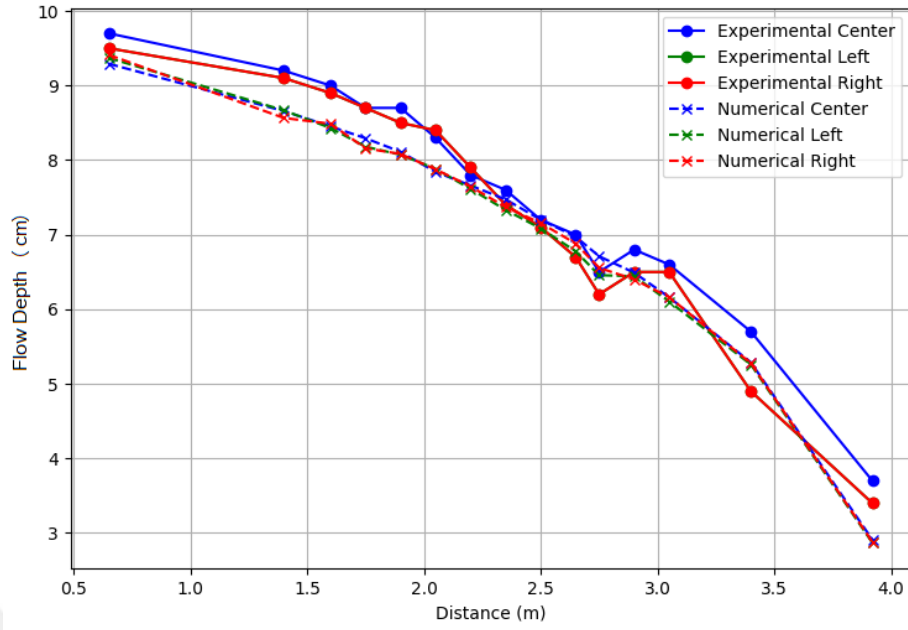


Figure 4.59. Variation of NN depth for rectangular arc obstacles $\dot{Q} = 0.004 \text{ m}^3/\text{s}$, $S=1.406^\circ$

Analysis of these data provides valuable insights into flow behavior under specific drainage and slope conditions. Through experimental and numerical measurements, differences in flow depth was observed, especially around the locations of rectangular obstacles. These differences indicate the influence of obstacles on flow dynamics, leading to changes in the depth distribution along the channel.

Comparison of experimental measurements with numerical simulations reveals reasonable agreement between the two data sets, despite notable discrepancies in some areas. This indicates that the numerical model effectively captures the complex flow behavior induced by obstacles. The consistency between experimental data and numerical predictions confirms the model's accuracy in simulating flow dynamics in channels with geometric complexity. However, the large discrepancy in error ratios highlights areas where it may be necessary to improve the numerical model to achieve a better fit with experimental observations. Given a discharge of $0.004 \text{ m}^3/\text{s}$ and a slope of 0.1406° , the discussion may delve into the implications for velocity distribution, flow line separation, vortex formation and shear stresses. The interplay between obstacle geometry, fluid properties, and flow dynamics plays a pivotal role in shaping these phenomena, highlighting the complex interaction within the channel environment.

In Fig. 4.60 for $\dot{Q} = 0.0045 \text{ m}^3/\text{s}$, $S=1.406^\circ$ the physical explanations are similar to the ones in Fig. 4.44. (Average error for left: 5.9858%, for center: 4.7027%, for right: 6.3247%)

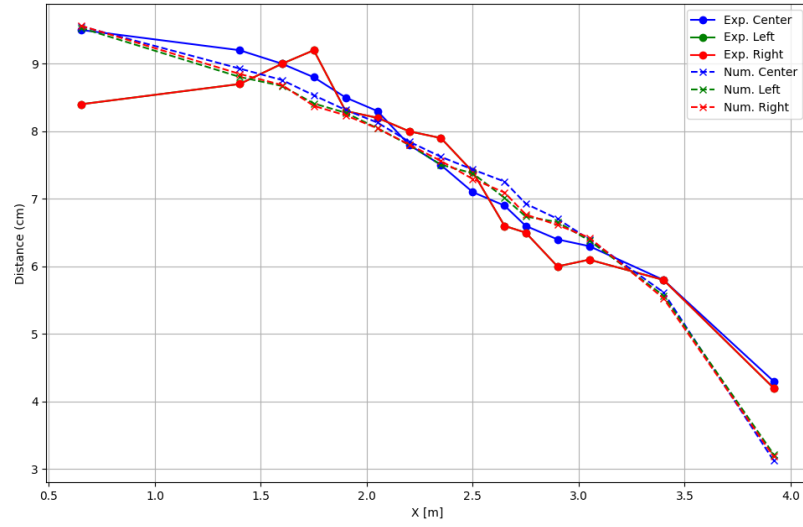
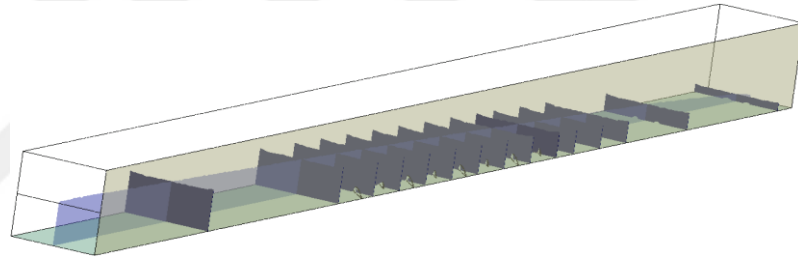
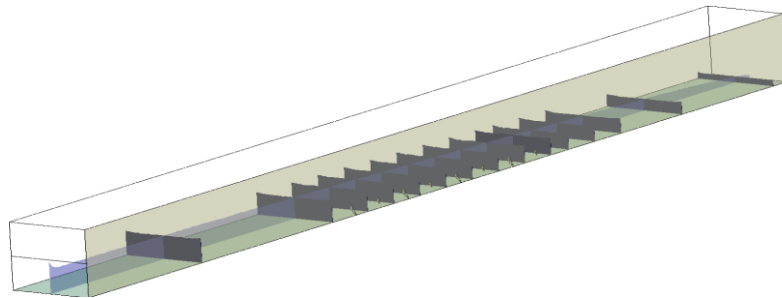


Figure 4.60. Variation of NN depth for rectangular arc obstacles at $\dot{Q} = 0.0045 \text{ m}^3/\text{s}$, $S=1.406^\circ$

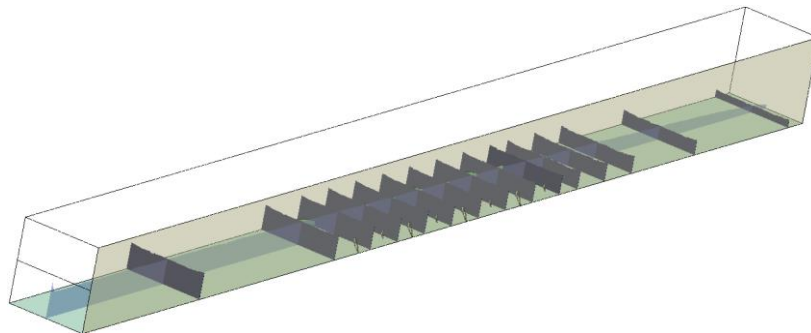
In Fig. 4.61, Fig. 4.62 and Fig. 4.63 there are some numerical results showing the depth of the bentonite.



a)



b)



c)

Figure 4.61. Depth profiles at slope **a)** $S = 0$, **b)** $S = 1.0156^\circ$ **c)** $S = 1.406^\circ$ for $\dot{Q} = 0.0035 \text{ m}^3/\text{s}$

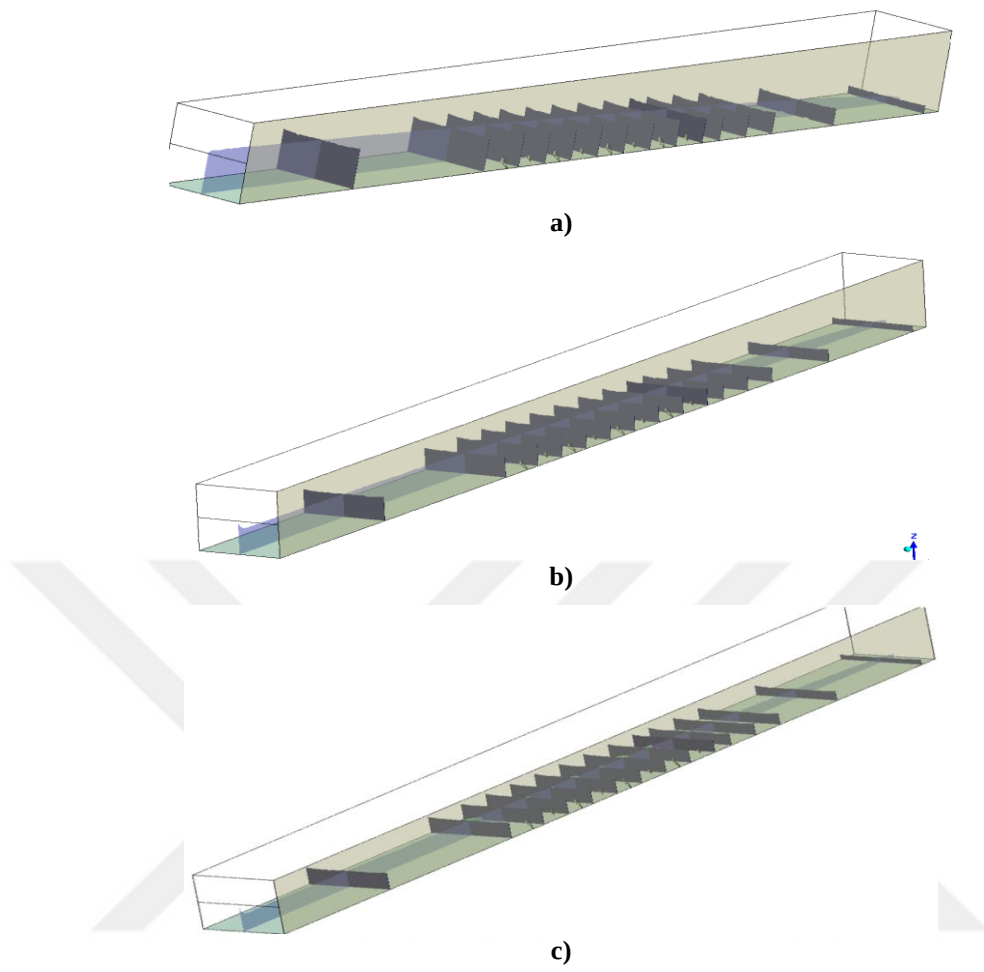


Figure 4.62. Depth profiles at slope **a)** $S = 0$, **b)** $S = 1.0156^\circ$ **c)** $S = 1.406^\circ$ for $\dot{Q} = 0.0040 \text{ m}^3/\text{s}$

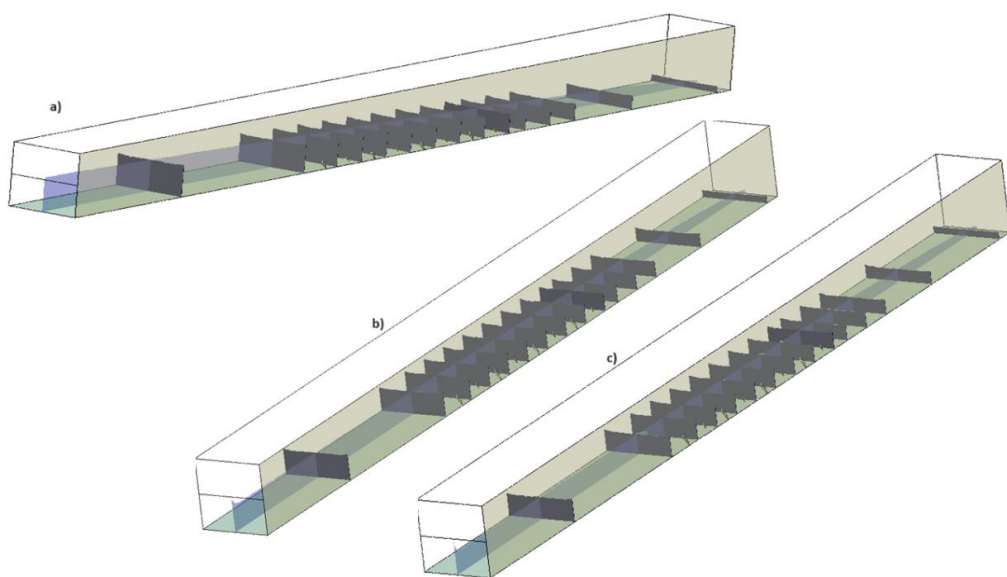


Figure 4.63. Depth profiles at slopes **a)** $S = 0$, **b)** $S = 1.0156^\circ$ **c)** $S = 1.406^\circ$ for $\dot{Q} = 0.0045 \text{ m}^3/\text{s}$

Some experimental photos of depth analysis are given below. In Fig. 4.64 and Fig. 4.65 there are some photos taken during the experiments showing the depth analysis definitely.

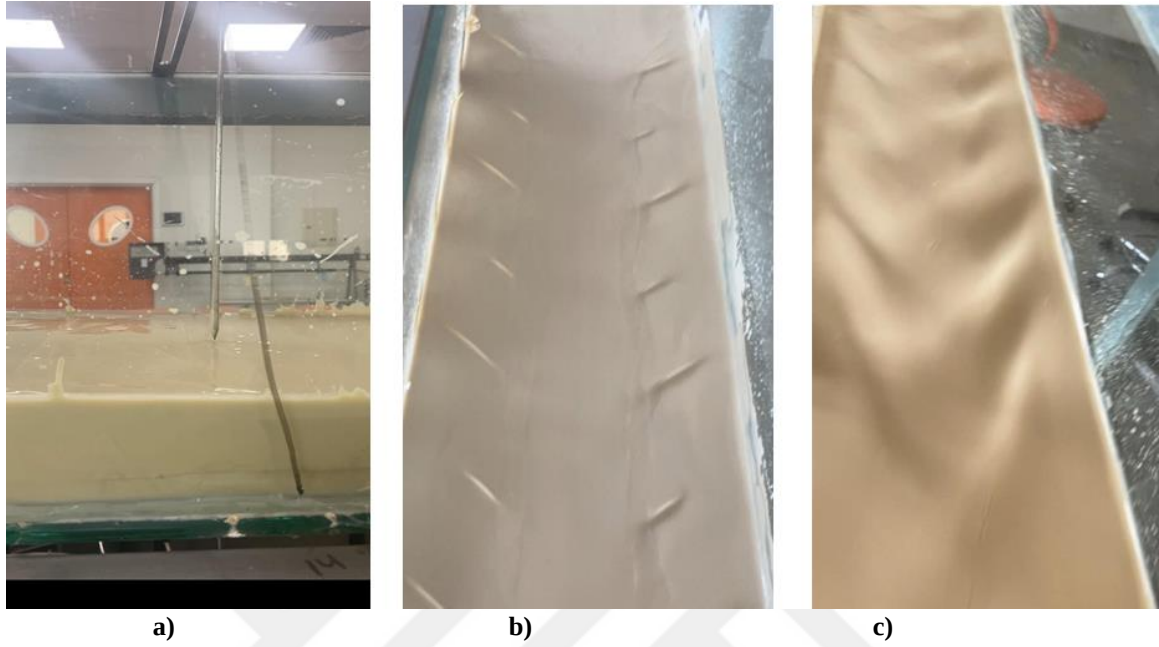


Figure 4.64. Photos from lab. **a)** Depth measuring **b)** bentonite after running prosses (at rest) **c)** bentonite starting running flow ($S = 1.406^\circ$ for $\dot{Q} = 0.0045 \text{ m}^3/\text{s}$)

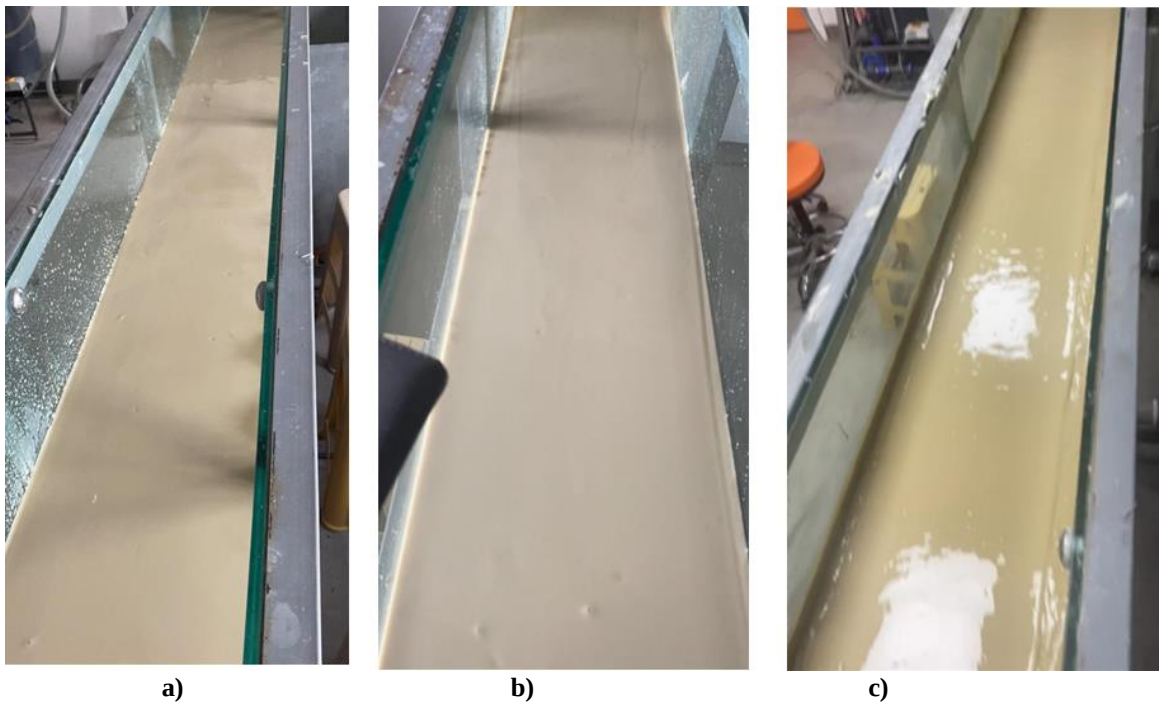


Figure 4.65. Top view profiles of bentonite in different slope at full running process with **a)** $S = 1.406^\circ$ **b)** $S = 1.0156^\circ$ **c)** $S = 0$, for $\dot{Q} = 0.0045 \text{ m}^3/\text{s}$

4.3.3. Depth Analysis of the Cube Obstacles

The presented dataset in Fig. 4.66 for $\dot{Q} = 0.0035 \text{ m}^3/\text{s}$, $S=0^\circ$ provides insights into the flow behavior of a non-Newtonian fluid inside a channel characterized by cubic-shaped obstacles. By examining experimental and numerical measurements, a comprehensive understanding of flow dynamics under specific drainage and slope conditions can be gained.

Analysis of experimental data reveals variations in flow depth, especially around the locations of cube obstacles. These differences highlight the influence of obstacles on flow behavior, affecting the depth distribution along the channel. Comparison of experimental measurements with numerical simulations indicates reasonable agreement between the two data sets. This indicates that the numerical model effectively captures the complex flow behavior induced by cubic obstacles, demonstrating its accuracy in simulating flow dynamics within channels characterized by geometric complexities. Given a discharge of $0.0035 \text{ m}^3/\text{s}$ and zero channel slope, the discussion may delve into the implications for velocity distribution, flow line separation, vortex formation and shear stresses. The interplay between obstacle geometry, fluid properties, and flow dynamics plays a crucial role in shaping these phenomena, underscoring the complex mechanisms at play within the channel environment. (Average error for left: 5.1631%, for center: 5.8092%, for right: 5.2580%)

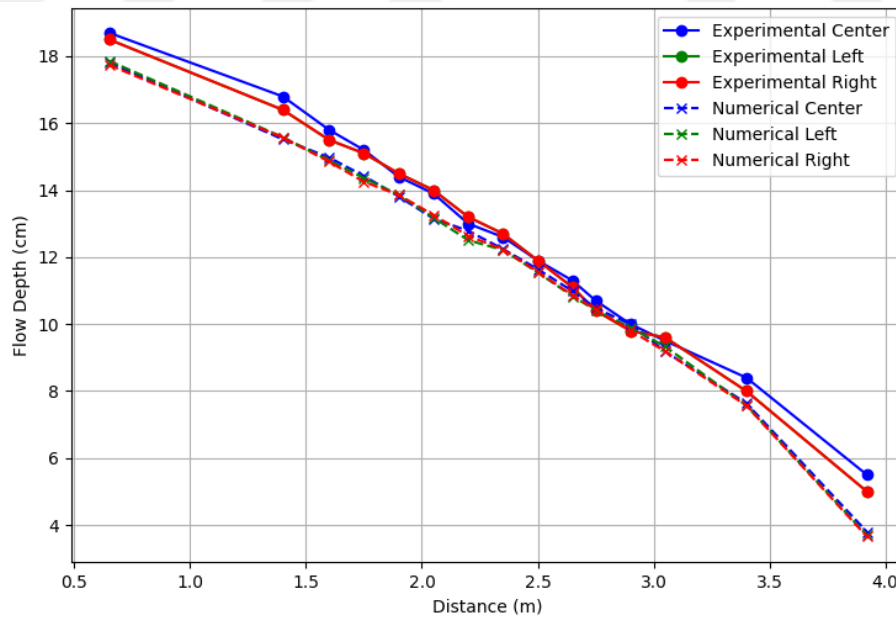


Figure 4.66. Variation of NN depth for cube obstacles at $\dot{Q} = 0.0035 \text{ m}^3/\text{s}$, $S=0^\circ$

The data presented in Fig. 4.67 for $\dot{Q} = 0.004 \text{ m}^3/\text{s}$, $S=0^\circ$ shows flow depth measurements. These measurements allow us to understand how the presence of obstacles affects flow behavior, especially in terms of depth distribution across the channel. Examining the experimental data, it has been also observed the differences in flow depth, with noticeable changes occurring near the locations of the cube obstacles. These variations indicate that obstacles affect the flow shape, leading to fluctuations in depth along the channel.

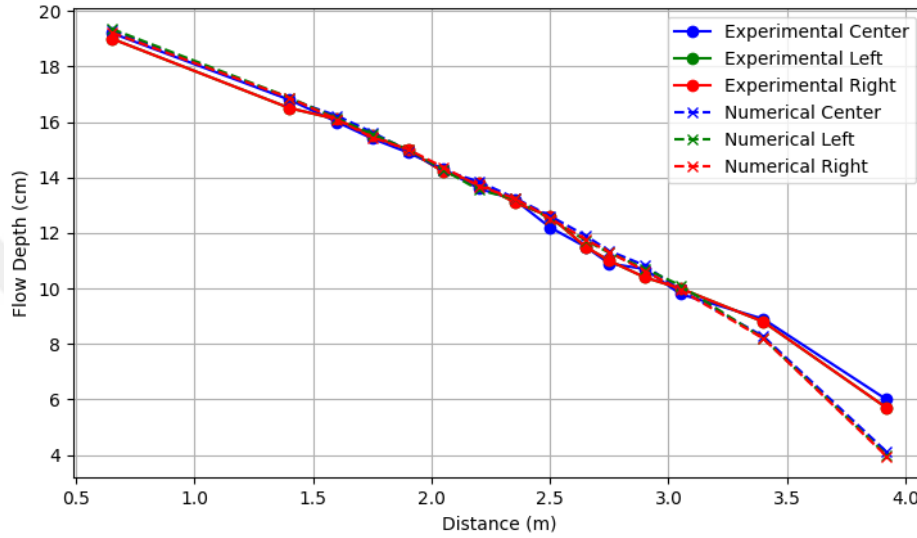


Figure 4.67. Variation of NN depth for cube obstacles at $\dot{Q} = 0.004 \text{ m}^3/\text{s}$, $S=0^\circ$

Differences in flow depth caused by cube obstructions have effects on the flow velocity distribution, boundary layer development, and shear stress distribution along the channel. These factors are critical for understanding sediment transport, scour, and erosion processes in natural and engineered channels. Overall, experimental and numerical data provide valuable insights into the flow behavior in channels with cubic obstacles under specific flow conditions. Understanding these dynamics is essential for various engineering and ecological applications, such as river engineering, flood management, and habitat restoration. (Average error for left: 3.6188%, for center: 3.9128%, for right: 3.5329%)

The data presented in Fig. 4.68 for $\dot{Q} = 0.0045 \text{ m}^3/\text{s}$, $S=0^\circ$ provides insights into the flow depth distribution in a cubic obstructed channel. Analysis of these measurements helps us understand how changes in flow conditions affect flow behavior, especially in terms of depth variation along the channel. When examining the experimental data, fluctuations in the flow depth was noticed, especially around the locations of the cube obstacles. These differences indicate that the presence of obstacles significantly affects the flow shape, leading to changes in the depth distribution. (Average error for left: 6.6365%, for center: 7.0576%, for right: 6.8046%)

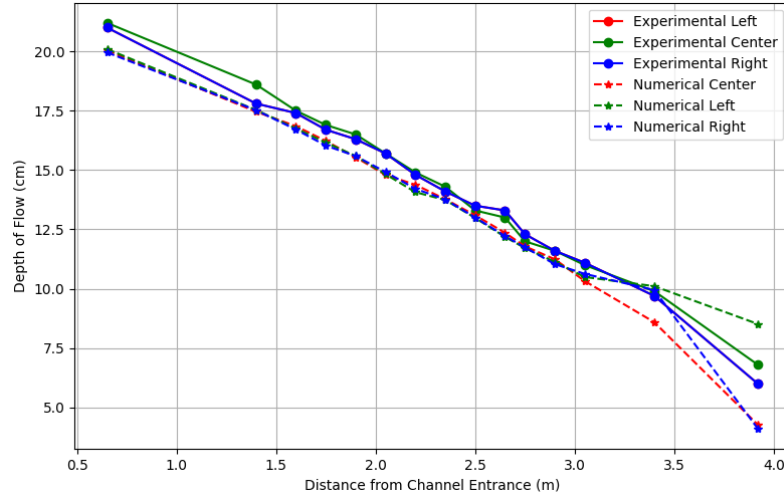


Figure 4.68. Variation of NN depth for cube obstacles at $\dot{Q} = 0.0045 \text{ m}^3/\text{s}$, $S=0^\circ$

In Fig. 4.69 for $\dot{Q} = 0.0035 \text{ m}^3/\text{s}$, $S=1.0156^\circ$ the experimental and numerical results depict the depth of flow for a non-Newtonian fluid flowing over cube obstacles within an open channel. The experimental data shows how the depth varies across different locations along the channel, while the numerical simulation provides insights into the expected depth based on computational models. Analyzing the experimental data, Depth of flow generally decreased as it moves downstream along the channel. This reduction in depth could be attributed to various factors, including energy losses due to friction and turbulence caused by the presence of obstacles. Additionally, variations in the geometry and spacing of the obstacles can lead to changes in flow behavior, resulting in fluctuations in depth. (Average error for left: 5.0027%, for center: 4.1410%, for right: 4.8925%)

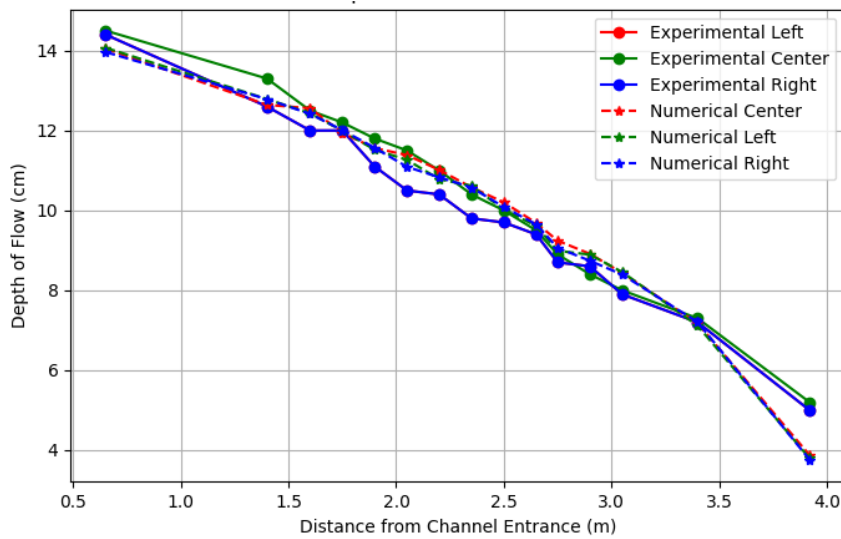


Figure 4.69. Variation of NN depth for cube obstacles at $\dot{Q} = 0.0035 \text{ m}^3/\text{s}$, $S=1.0156^\circ$

For the scenario shown in Fig. 4.70, for $\dot{Q} = 0.004 \text{ m}^3/\text{s}$, $S=1.0156^\circ$ the experimental data reveals a constant trend of decreasing flow depth as it moves downstream along the channel.

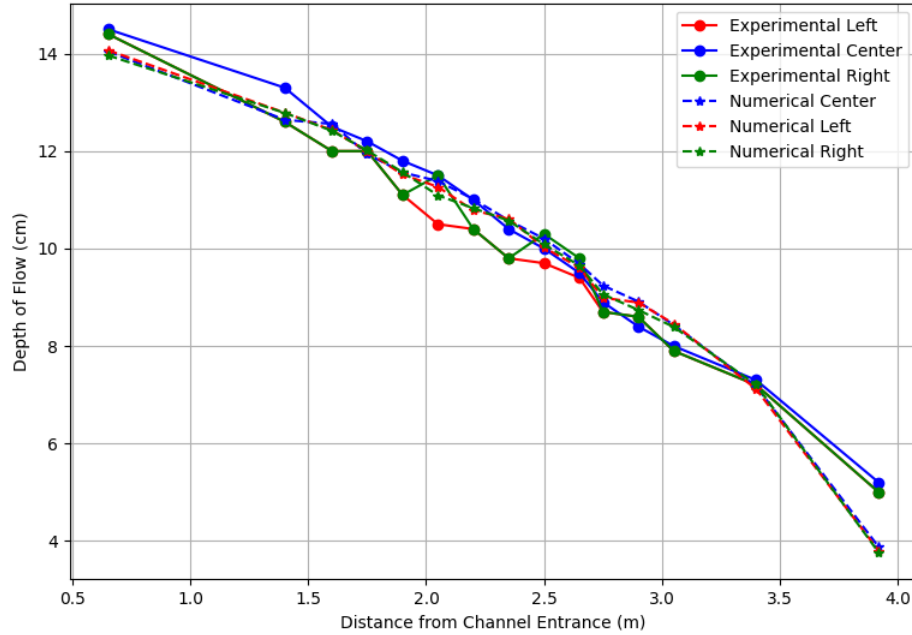


Figure 4.70. Variation of NN depth for cube obstacles at $\dot{Q} = 0.004 \text{ m}^3/\text{s}$, $S=1.0156^\circ$

The decrease in depth can be attributed to various factors such as energy dissipation due to turbulence and friction loss caused by the presence of obstacles. In addition, differences in channel geometry and obstacle spacing contribute to fluctuations in the depth profile.

Comparing experimental observations with numerical simulations, reasonably a good agreement overall was found. The numerical model depicts the general trend of decreasing depth along the channel, which is in good agreement with experimental data at most sites. However, some discrepancies between the two data sets are noticeable, especially at specific points along the channel. These differences may arise from simplifications or assumptions made in the numerical model, as well as uncertainties in the experimental measurements. In summary, the results confirm the complex interplay between obstacle geometry, fluid properties, and flow dynamics within the channel environment. Improving and validating numerical models against additional experimental data can enhance our understanding of the fundamental mechanisms governing flow behavior in such scenarios. (Average error for left: 3.3402%, for center: 4.0462%, for right: 3.6780%).

As is evident in both experimental and numerical data given in Fig. 4.71, for the case of $\dot{Q} = 0.0045 \text{ m}^3/\text{s}$, $S=1.0156^\circ$ the flow depth generally decreases with increasing distance from the channel inlet. This trend is typical in open channel flow, where the hydraulic properties of the

channel affect the flow depth distribution along its length. Overall, there is reasonable agreement between the experimental and numerical data, with similar trends observed in the depth distribution. However, some inconsistencies exist, which are particularly noticeable in certain sections of the channel where numerical predictions deviate from experimental measurements. These discrepancies can be due to simplifications or assumptions made in the numerical model, differences in boundary conditions, or uncertainties in experimental measurements. Understanding flow depth distribution is crucial for various engineering applications, such as channel design, flood risk assessment, and water resources management. Insights gained from this analysis can inform decisions regarding channel engineering, sediment transport, and design of hydraulic structures to improve channel performance and reduce potential risks. Improving the numerical model, calibrating, and validating it against additional experimental data can enhance its accuracy and reliability. Long-term monitoring and analysis of channel flow dynamics can provide valuable insights into temporal and seasonal variations in flow behavior. Incorporating additional factors such as sediment transport, turbulence, and boundary roughness can lead to a more comprehensive understanding of channel hydraulics and improve the predictive capabilities of numerical models. In conclusion, the presented analysis provides valuable insights into the flow depth distribution within the channel, highlighting the convergence and divergence between experimental observations and numerical predictions. This knowledge contributes to the development of hydraulic modeling and informs decision-making processes in water resources management and engineering design. (Average error for left: 5.8610%, for center: 5.2044%, for right: 5.8729%)

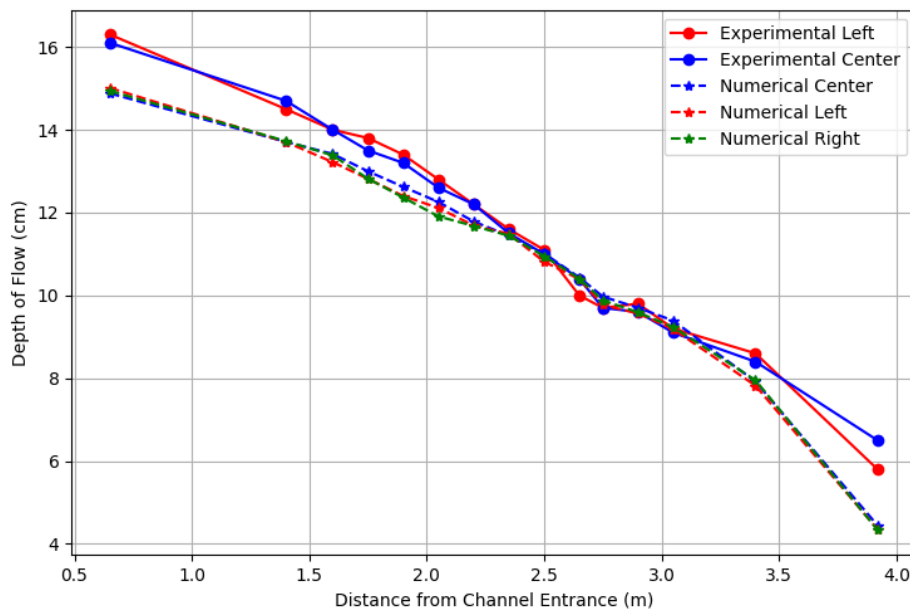


Figure 4.71. Variation of NN depth for cube obstacles at $\dot{Q} = 0.0045 \text{ m}^3/\text{s}$, $S=1.0156^\circ$

For the case of $\dot{Q} = 0.0035 \text{ m}^3/\text{s}$, $S=1.406^\circ$ Fig. 4.72 shows a general trend of decreasing flow depth with increasing distance from the channel entrance, consistent with open channel flow expectations. However, there are inconsistencies between experimental and numerical data, especially in certain sections of the channel. These differences may arise from various factors such as simplifications in the numerical model, differences in boundary conditions, or uncertainties in experimental measurements. Understanding flow depth distribution is critical for applications such as channel design, flood risk assessment, and water resource management. Insights gained from this analysis can inform decisions regarding channel geometry, sediment transport, and hydraulic structure design to improve channel performance and mitigate potential risks. Improving the numerical model and validating it against additional experimental data can improve its accuracy and reliability. Long-term monitoring and analysis of channel flow dynamics can provide valuable insights into temporal and seasonal variations in flow behavior. Incorporating additional factors such as sediment transport, turbulence, and boundary roughness can enhance the comprehensiveness of the numerical model and improve its predictive capabilities. In summary, this analysis contributes to the understanding of flow depth distribution in the channel and highlights the convergence and divergence between experimental observations and numerical predictions. These ideas advance hydraulic modeling and support decision-making processes in water resources management and engineering design. (Average error for left: 5.3986%, for center: 5.9646%, for right: 5.4020%)

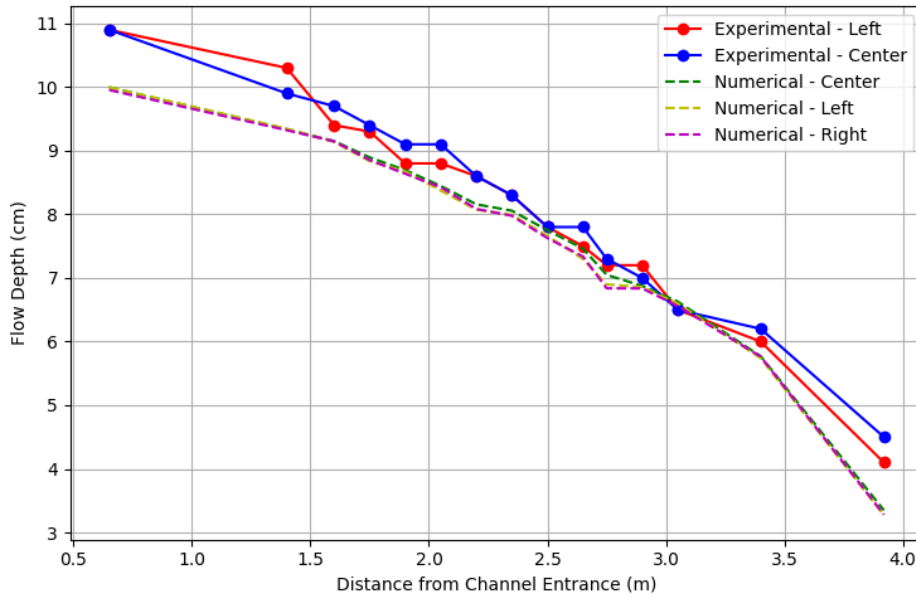


Figure 4.72. Variation of NN depth for cube obstacles at $\dot{Q} = 0.0035 \text{ m}^3/\text{s}$, $S=1.406^\circ$

The plotted data for $\dot{Q} = 0.004 \text{ m}^3/\text{s}$, $S=1.406^\circ$ in Fig. 4.73 represents the flow depth distribution along the channel for both experimental and numerical simulations. Overall, the numerical model closely resembles the trends of the experimental data, indicating a reasonable agreement between the two. Both show a gradual decrease in flow depth from the channel inlet to the downstream end, which is typical for open channel flows. (Average error for left: 17.4010%, for center: 19.6185%, for right: 17.2890%)

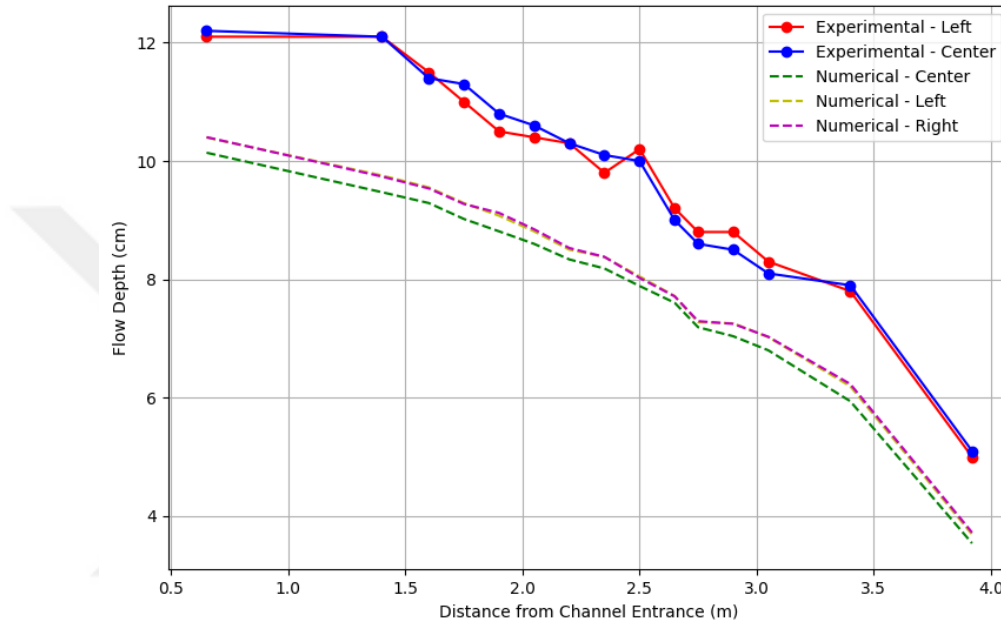


Figure 4.73. Variation of NN depth for cube obstacles at $\dot{Q} = 0.004 \text{ m}^3/\text{s}$, $S=1.406^\circ$

Data for $\dot{Q} = 0.0045 \text{ m}^3/\text{s}$, $S=1.406^\circ$ given in Fig. 4.74 show trends similar to the previous cases, with the flow depth gradually decreasing along the channel. Both experimental and numerical results show this behavior, indicating a consistent pattern. However, there are some discrepancies between the experimental data and the numerical data, especially in the center section and lower part of the channel. These differences can arise from various factors, such as uncertainty in experimental measurements, simplifications in the numerical model, or limitations in boundary condition settings. Despite these discrepancies, the general agreement between the two data sets indicates that the numerical model can reasonably predict the flow depth distribution under these conditions. Further optimization and validation efforts are necessary to improve the accuracy of the model and expand its application in practical engineering scenarios. (Average error for left: 6.0955%, for center: 7.2848%, for right: 5.1926%)

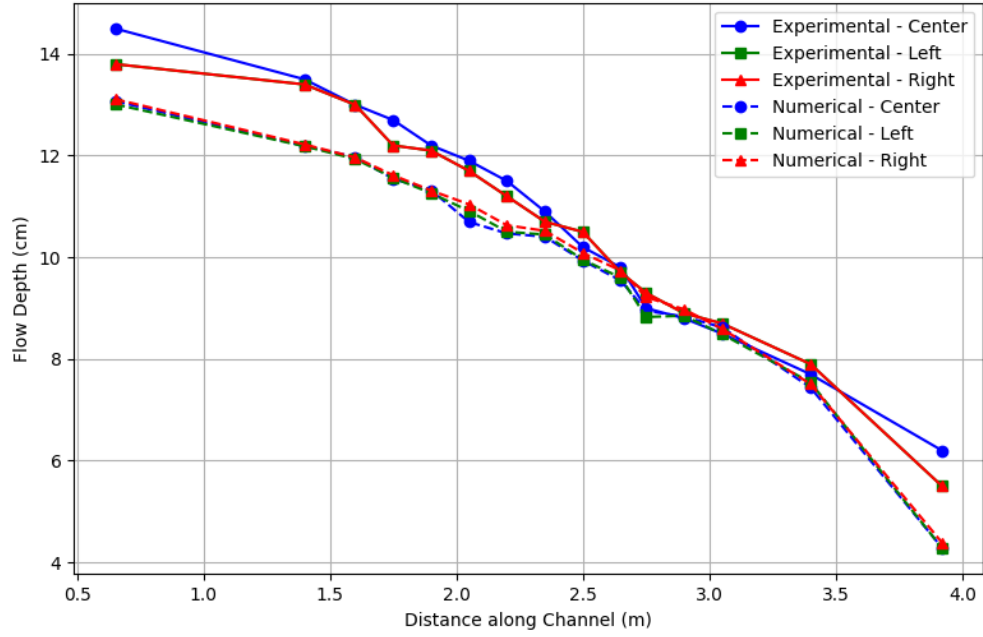


Figure 4.74. Variation of NN depth for cube obstacles at $\dot{Q} = 0.0045 \text{ m}^3/\text{s}$, $S=1.406^\circ$

4.3.4. Depth Analysis of the Cylindrical Obstacles (Experimental only)

As it can be seen from Fig. 4.75, Fig. 4.76 and Fig. 4.77 for all flowrates and slopes examined similar trends were observed. At a slope of 0° , the flow depth remained relatively constant across the channel, indicating a stable hydraulic system with balanced flow dynamics. Minimal difference in depth from left to right indicates a uniform flow distribution. With a slope of 1.0156° , a slight increase in flow depth was observed compared to a slope of 0° . The depth variation becomes more pronounced from left to right, indicating a change in flow dynamics due to increasing slope. This increase in slope resulted in slightly higher velocities, which increased depth toward the right side of the channel. At the steeper slope of 1.406° , the flow depth showed a significant increase compared to the shallower slopes. The depth variation becomes more pronounced from left to right, indicating accelerated flow dynamics due to the steeper slope. Steeper slopes resulted in higher velocities, resulting in a more pronounced depth difference across the channel.

In general, increased discharge and slope angle resulted in higher flow velocities, which resulted in increased flow depth, especially toward steeper sections of the channel. Steeper slopes resulted in sharper depth differences across the channel, indicating more accelerated flow dynamics.

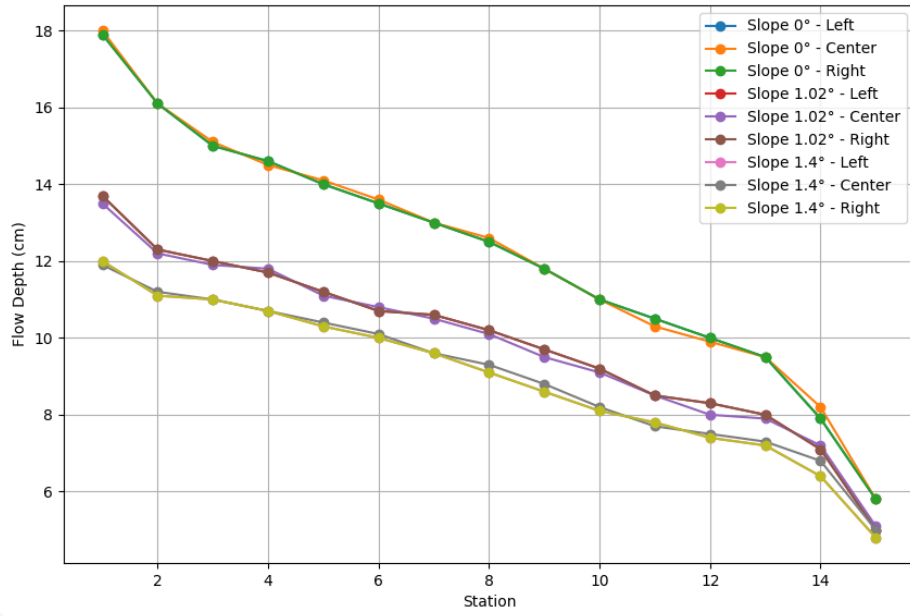


Figure 4.75. Variation of NN depth for cylindrical obstacles at $\dot{Q} = 0.0035 \text{ m}^3/\text{s}$

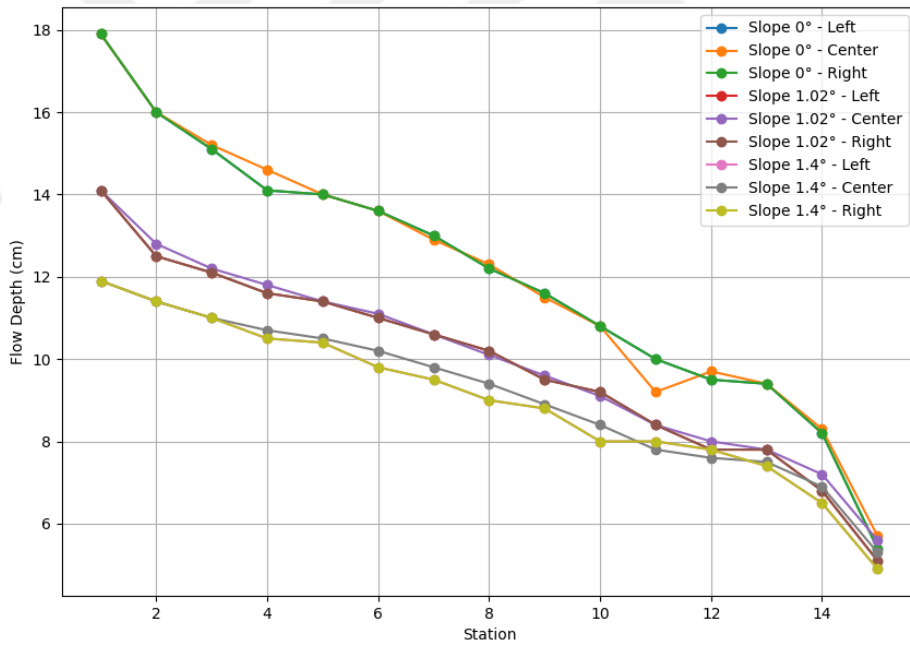


Figure 4.76. Variation of NN depth for cylindrical obstacles at $\dot{Q} = 0.004 \text{ m}^3/\text{s}$

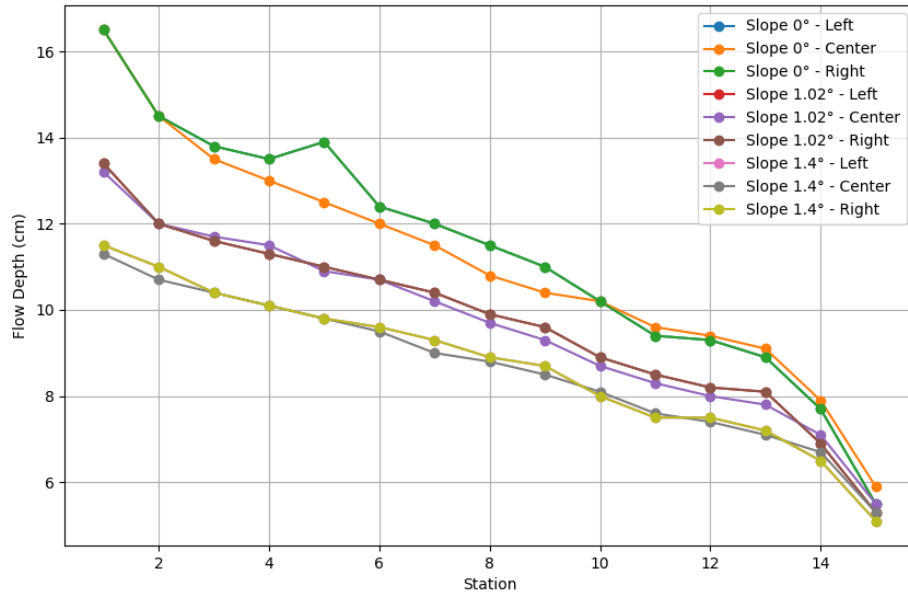


Figure 4.77. Variation of NN depth for cylindrical obstacles at $\dot{Q} = 0.0045 \text{ m}^3/\text{s}$

4.3.5. Depth Analysis with Elliptical Obstacles (Experimental only)

The flow depth data for the three different slopes (0° , 1.0156° and 1.4°) with a constant discharge of $0.0035 \text{ m}^3/\text{s}$ show notable trends as seen in Fig. 4.78. As the slope increases, the flow depth generally decreases across all sections (left, middle, right). This inverse relationship is expected because a steeper slope allows water to flow more quickly, reducing depth. At a 0° slope, flow depths are uniform and relatively higher, indicating slower flow that allows more water to accumulate at each station. As the gradient increases to 1.0156° and then to 1.406° , the depths decrease similarly. This is most evident in the middle section, which consistently shows slightly higher depths compared to the left and right sections across all slopes, likely due to channel geometry and flow dynamics that favor central accretion. The data indicate that increasing slope leads to enhanced flow velocities and decreased depths, which is consistent with the principles of fluid dynamics in open channels. Understanding these changes is critical to designing and managing waterways, ensuring efficient flow, and preventing problems such as flooding or erosion.

In Fig. 4.79 the flow depth data for the three different slopes (0° , 1.0156° and 1.406°) with a constant discharge of $0.004 \text{ m}^3/\text{s}$ show trends similar to those observed with $0.0035 \text{ m}^3/\text{s}$. For 0° slope, flow depths are highest, indicating that water is moving slower and accumulating more at each station. The left, middle, and right flow depths are quite similar, showing a uniform distribution across the channel. When channel slope increased to 1.0156° , the flow depths decrease compared to a 0° slope. The dip remains constant across all sections, reflecting a slightly steeper slope that allows water to flow more quickly, reducing accumulation. For 1.406° degree of

slope, the flow depths are the lowest among the three slopes. The significant decrease in depth highlights the effect of the steeper slope, which accelerates the flow, resulting in less water accumulating at each station. The consistent pattern across both discharges ($0.0035 \text{ m}^3/\text{s}$ and $0.004 \text{ m}^3/\text{s}$) shows that an increase in slope leads to higher flow velocities and thus lower flow depths. This behavior is consistent with the principles of fluid dynamics in open channels, where steeper slopes facilitate faster water movement, reducing depth. Understanding these trends is essential for designing effective water conveyance systems and managing flow rates to prevent problems such as erosion or flooding.

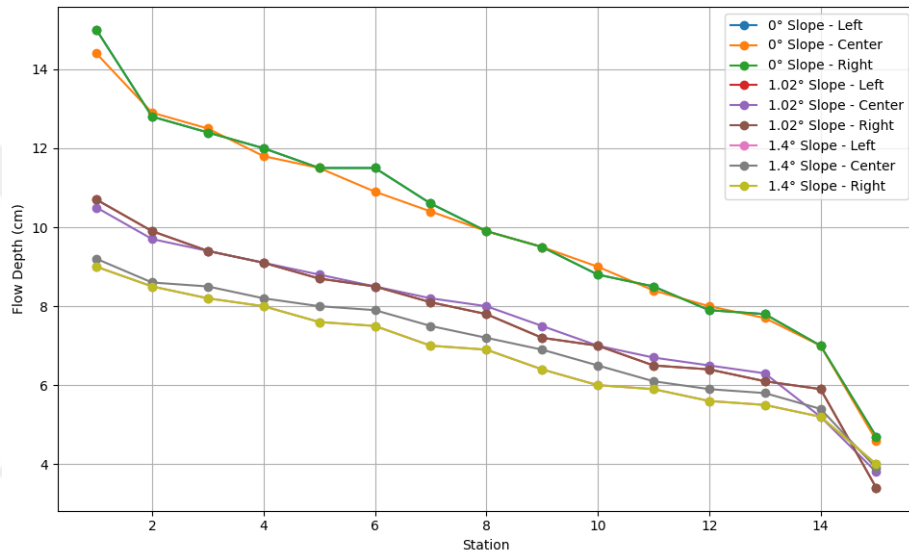


Figure 4.78. Variation of NN depth for elliptical cylindrical obstacles at $\dot{Q} = 0.0035 \text{ m}^3/\text{s}$

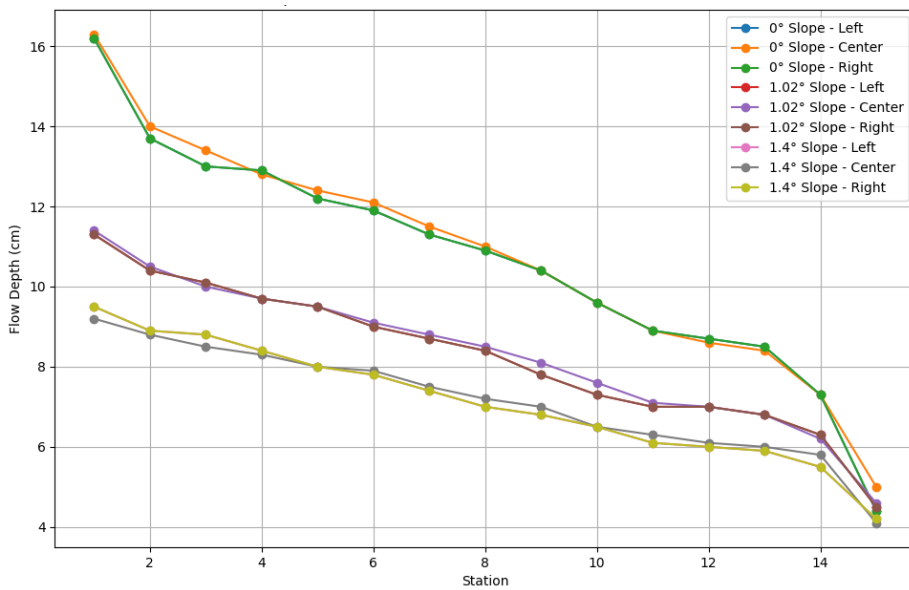


Figure 4.79. Variation of NN depth for elliptical cylindrical obstacles at $\dot{Q} = 0.004 \text{ m}^3/\text{s}$

In Fig. 4.80, the flow depth data for the three different slopes (0° , 1.02° , and 1.4°) with a constant discharge of $0.0045 \text{ m}^3/\text{s}$ show trends similar to those observed with low flowrates. The 0° slope shows the highest flow depths, indicating slower water flow and greater accumulation at each station, with uniform flow depths on the left, middle, and right across the channel. As the slope increases to 1.02° , flow depths decrease relative to the 0° slope, demonstrating the effect of a slightly steeper slope facilitating faster water movement and less accumulation. A slope of 1.4° , being the steepest, results in the lowest flow depths, reflecting the fastest flow and minimal accumulation at each station. This consistent pattern across different discharges and slopes highlights basic principles of fluid dynamics in open channels. Increasing the slope results in higher flow velocities and lower flow depths due to less water accumulation at each station. This understanding is critical to the design and management of water conveyance systems, ensuring efficient flow and preventing problems such as erosion or flooding.

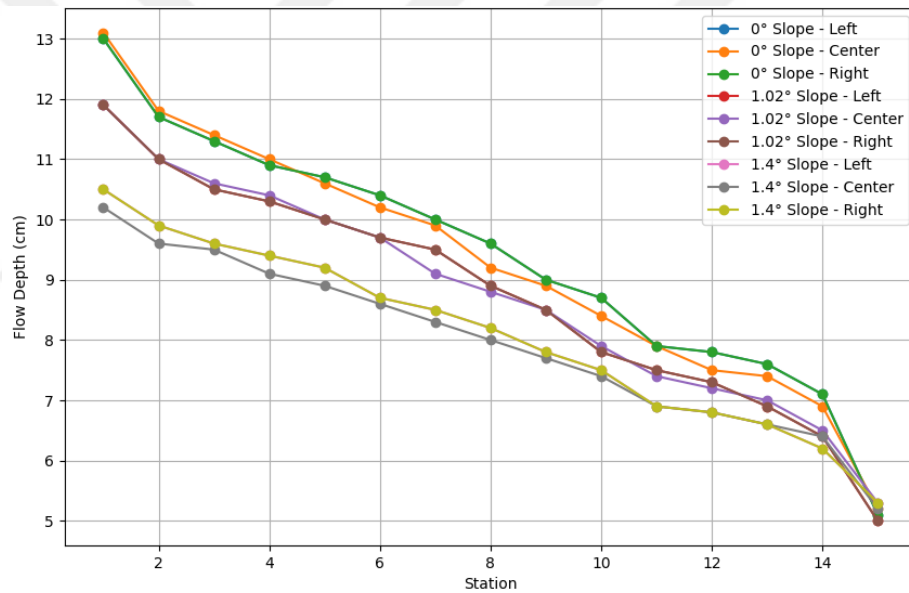


Figure 4.80. Variation of NN depth for elliptical cylindrical obstacles at $\dot{Q} = 0.004 \text{ m}^3/\text{s}$

4.4. Comparison of All Obstacles by Means of Depth Analysis

For the bentonite mixture flow rate inside the channel, comparisons have been made for the velocity changes in the center between all obstacles for all cases of different flows and slopes as it has been shown in Fig. 4.81, Fig. 4.82 and Fig. 4.83.

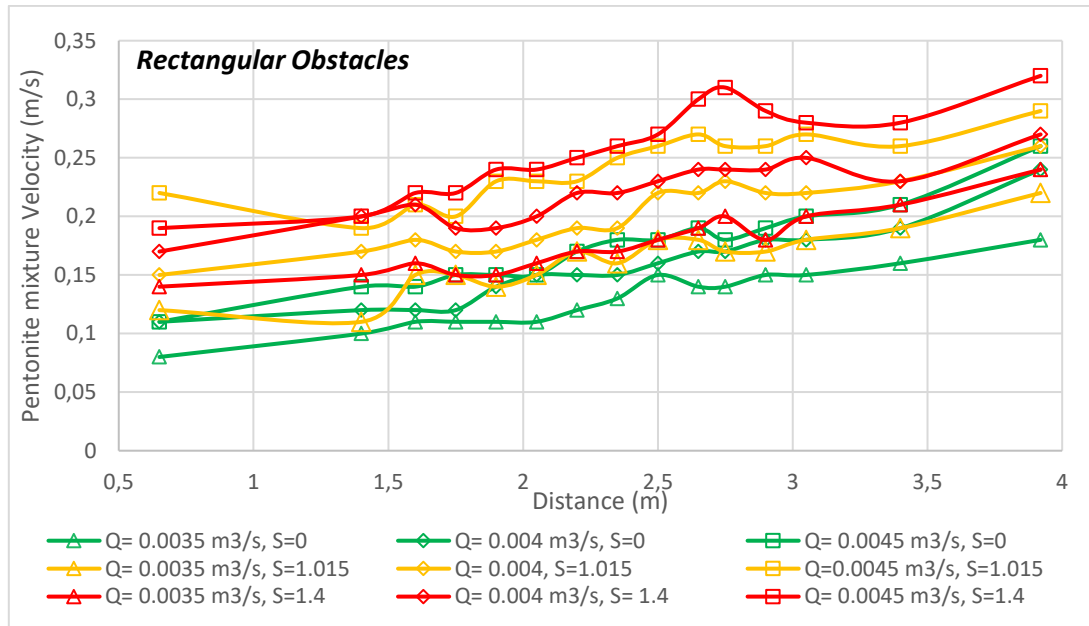


Figure 4.81. Experimental velocities for bentonite mixture in the center of the channel for rectangle obstacle

Figure 4.81 shows that the velocity values for all cases are raising along the channel with a relatively steady rate. The velocity downstream increases about 1.5 to 2.6 of the upstream values. For rectangular obstacle with no slope condition, the velocity after the obstacles raised in comparison to the velocity before the obstacles with a factor of 2.2.

For higher flow rates at 0 slope, the velocity raised, but with slightly less steep. At flow of $0.0045 \text{ m}^3/\text{s}$ and 1.4% slope, the velocity difference decreases from 0.19 to 0.32 m/s. The acceleration of the stream decreases as the flow and slope increases, because the depth of the fluid increase over the obstacle height and reduces obstacle's effect on the bulk of the fluid. Consequently, the flow separation rate and size around the obstacles decreases for relatively high viscosity of the bentonite.

At flow rate of $0.0045 \text{ m}^3/\text{s}$ and slope of 1.406 deg, the velocity surges before the end of the obstacle zones in contrast to the water velocity profile. The main reason is that the extend of the effect of the velocity variation at very after of the last obstacle for such viscous fluid flow.

For arc obstacle, shown in Fig. 482 similar pattern of velocity growth repeated. However, the differences of velocity rise between the upstream and downstream for all cases of flow and slopes are quite comparable. The reason behind that is the height of the obstacle changes in a defined curve with higher values at the middle and 0 cm at the walls and near the center of the channel. This suggests more flow separations than the other types of obstacles at both sides and the middle and it implies that its effects are more robust.

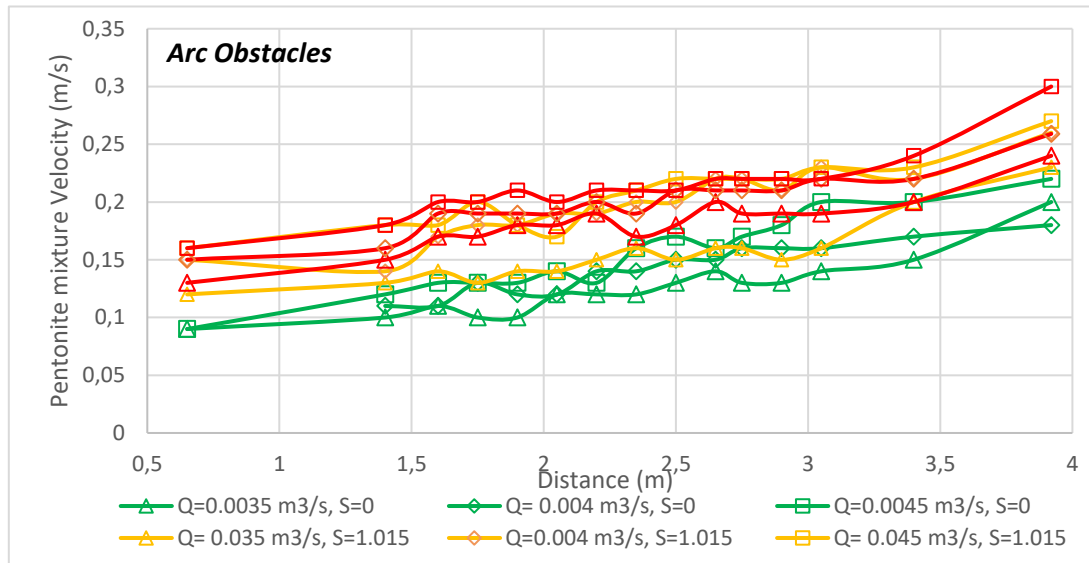


Figure 4.82. Experimental velocities for bentonite mixture in the center of the channel for rectangular arc obstacle

For cubic obstacle as shown in Fig. 4.83, the velocity gradient along the channel is semi-similar at various flow and slopes. but the rate of velocity changes is sharper in contrast to the rectangular obstacles. The separation observed clearer after the obstacles in comparison to the rectangular type, because of the spaces between the obstacles. These spaces will give the fluid to escape more smoothly and encourages different localized velocity values.

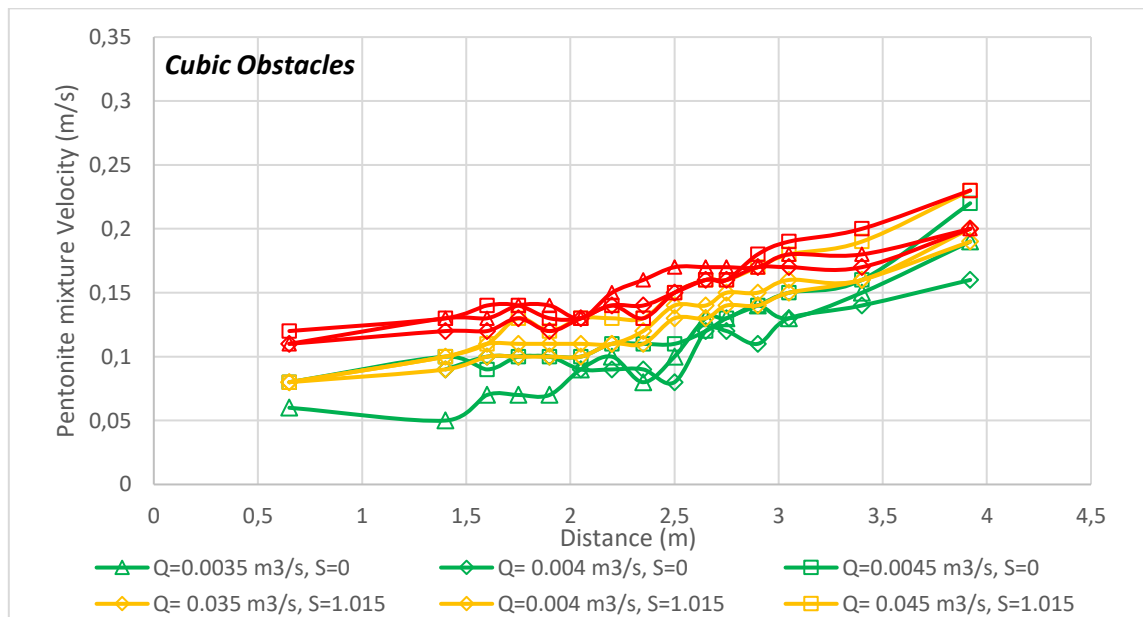


Figure 4.83. Experimental velocities for bentonite mixture in the center of the channel for cubic obstacle

There are flow separations and even sediments at the bottom of the flow attached to the bed, which shows velocity differences of the flow, Fig. 4.84. These differences are noticeable for all cases and the velocity tends to be zero after the upstream obstacles. As the flow goes further through obstacles, the sedimentation decreases because of stronger flow separation and high velocity.

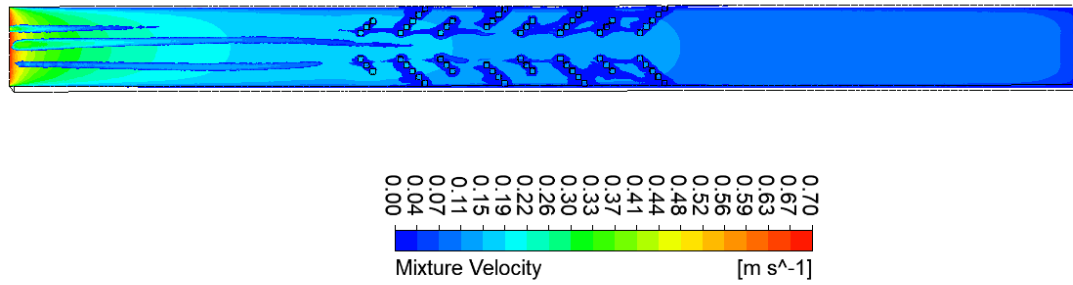


Figure 4.84. Velocity profile at the bottom of the bentonite mixture at $Q=0.0035 \text{ m}^3/\text{s}$, slope = 0 for cubic obstacles.

The height of the obstacles in comparison to the depth of the fluid is low. For high flow rates, where both velocity and fluid depth increases, the effect of the obstacles on the velocity decrease due to its low holdback effects on the bulk flow of the fluid.

As it is shown in the curves and for all cases, the velocity rise fluctuated at the region of the obstacles and this demonstrate the effects of the them. Despite that, the effect of the obstacles is less than in case of water flow because of the high viscosity of the bentonite and higher difference between the bentonite mixture depth relative to the obstacles. Higher obstacles will add further local resistances to the flow and will persuade additional flow separation intensity. Nevertheless, the obstacles have regular tougher and wider effects on the velocity of the bentonite mixture before the obstacles in contrast to the water flow inside the channel, mainly because of its higher viscosity and density.

4.5. Comparison to the Literature

The work in Gond [14] highlights distinct focuses and methodologies. Present study investigates the effects of obstacles on velocity profiles, turbulence generation, shear stress, and sediment transport in both Newtonian and non-Newtonian fluids. It uses simulations and experiments to monitor these behaviors, focusing on speed changes and the effects of different obstacle arrangements. In contrast, Gond [14] focuses on hydraulic jump separation in supercritical open channel flows, using an analytical model of supersonic aerodynamics to predict the hydraulic jump separation length. This study discusses the formation and properties of

discrete hydraulic jumps and their similarity to shock waves in gas flows, using experiments and numerical simulations to study flow patterns under these conditions.

Likewise, a comparison with Young [18] reveals different priorities. The current study delves into the fluid dynamics of Newtonian and non-Newtonian fluids in channels with obstacles, analyzing velocity profiles, turbulence, shear stress, and sediment transport. It combines experimental and computational fluid dynamics (CFD) simulations to achieve its goals. On the other hand, Young [18] focuses on thermal effects in a standard airflow duct with heated baffles, measuring heat transfer properties such as average Nusselt numbers and surface temperature differences through experimental investigations and numerical models. This study found that changes in duct height, heater power, and air flow rate significantly affect thermal parameters and determine the effect of obstacle arrangement on heat transfer. While both studies explore fluid-obstacle interactions, they differ in their primary focus, with the current study focusing on fluid dynamics and the other focusing on thermal effects.

Comparison of the current study with Liao and Chen [58] highlights the unique methods and implications. The present study explores fluid flow behavior within obstructed channels, using both experimental observations and CFD simulations to understand velocity profiles, turbulence generation, shear stress, and sediment transport. Their results are relevant for applications in hydraulic engineering and wear control. In contrast, Liao and Chen [58] studies the thermal effects of obstructions in airflow channels, and analyzes the heat transfer characteristics through experimental and numerical methods. The study focuses on how changes in duct height, heater power, and air flow rate affect heat transfer coefficients, with significant implications for thermal management, especially in electronic devices.

When comparing the present study with Riviere et. al [68], different phenomena in open channel flows seen by focusing. The present study investigates the effects of obstructions on velocity, turbulence, shear stress and sediment transport profiles for different types of fluids. Riviere et al. [68] Study open channel supercritical flow encountering obstructions, observing discrete hydraulic jumps and jet wall-like bow waves. They conduct experiments to determine the conditions for these phenomena and develop a conceptual model to explain the transition between them, with additional insights into water depth oscillations and flow-induced forces on structures. Both studies emphasize the importance of understanding flow behavior near obstacles, but the current study provides a broader analysis of fluid dynamics while Rivière et al. Focus on specific hydraulic phenomena.

Finally, a comparison with Christodoulou [69] shows different emphases on flow dynamics. The present study examines the velocity, turbulence, and shear stress profiles in channels with obstacles, especially for Newtonian and non-Newtonian fluids, using experimental and CFD simulations. It highlights increased flow separation and turbulence around obstacles.

Christodoulou [69] focuses on flow resistance over artificial roughness elements immersed in an inclined channel, evaluating different obstacle shapes to determine the effective Manning coefficient and equivalent sand roughness. While the current study delves into the detailed fluid dynamics affected by obstructions, Christodoulou [69] work focuses on how obstruction shape, size, and density affect flow resistance.

In summary, the present study provides a comprehensive analysis of fluid dynamics in obstructed channels and provides valuable insights into hydraulic engineering and sediment transport. It complements the results of other studies by focusing on different aspects of obstacle-fluid interactions, such as thermal effects, hydraulic phenomena, and flow resistance.



5. CONCLUSIONS

This section provides conclusions that have been made throughout this project and recommendations for further work. The aim and objectives of this thesis have been defined to fill some gaps found in the available materials through the literature review. Then, experimental and CFD simulations were completed for the flow of Newtonian and non-Newtonian fluids inside channels where obstacles existed based on the methodology that has been illustrated in Chapter 3. The computed results have been analyzed and compared against the experimental data. The following conclusions may be drawn:

- ✓ Regardless of being Newtonian or not, increasing the channel and flow of the liquid will consequently increase the effect of the obstacles on the velocity profile due to the growth of the flow separation and intensity around the obstacles.
- ✓ For both Newtonian and non-Newtonian tested fluids near the obstacle, velocity gradients are steeper, indicating accelerating and decelerating flow patterns. This will lead to insert local turbulence and swirling which can be observed more intensely behind the obstacles as it is expected in channel flow. The simulation and experiment results were compatible and confirmed such kind of fluid behavior.
- ✓ For both Newtonian and non-Newtonian tested fluids, since there is a space between obstacles starting from both sides of the channel toward the center, the flow results in increased shear stress along the channel bottom and walls. This affects sediment transport and erosion processes. One should consider this effect for selecting and surface treatment of the type of channel material.
- ✓ When examining the water flow, at 0° slope, the velocity of the fluid will continue to remain high after the obstacles on the flatbed. The velocity after the obstacles is about 1.2 to 1.8 times the velocity before obstacles compared with less liquid depth inside the channels. This phenomenon will help to reduce thermal energy transfer between the liquid and the surroundings due to a decrease in the surface area for heat exchange and less time of exposure to different boundary thermal conditions. However, when there is a slope of the bed greater than 0° , the velocity increase of the water after the obstacles drop, in some cases, the velocity is less than the velocity before the obstacles. The main reason is that the gravity force is dominant over the effect of the obstacles, especially for high mass flow rates. A higher mass flow rate means higher gravitational force inserting on the bulk.
- ✓ The square obstacle arrangement shows higher velocity change and higher velocity of the water at the flatbed after obstacles since it induces further swirls in different directions on the fluid. This lessens the effect of the boundary conditions at the bottom of the channel.

- ✓ The water velocity at the side walls is affected highly by different flow rates and obstacle types and arrangements. The main reason comes from the intensity and direction of the swirling on the boundary layer thickness and pressure gradients at different locations alongside the channel where the obstacles exist. The velocity close to the wall is decreased when the channel has slopes more than 0° because of the increasing gravity force of the bulk.
- ✓ The velocity and depth variation of the non-Newtonian fluid is less than water because of higher shear stress and different flow separations in the region of the obstacles. Furthermore, the rate of the change of the velocity is much smoother than the water throughout the channel.
- ✓ The relatively higher flow separation and effective boundary condition at the wall side of the channel leads to presenting a developed difference between the velocities at the center and close to the sidewalls. This suggests that the obstacles induce acceleration of the flow, leading to higher velocities in the central region and lower velocities near the obstacles' edges. The flow characteristics close to the wall on both sides are almost similar.

RECOMMENDATIONS

During the work on this project and based on the outcomes of this study, the following aspects need further study:

- ✓ Investigating flow characteristics for different shapes, height and arrangement of the obstacles to optimizing the finest outcomes for different fluids at various flows.
- ✓ Studying the possibilities of using obstacles in channels as pre-mixing means.
- ✓ Using different obstacles in the same channel to maximize the velocity of the fluid behind them.
- ✓ Repeating the experiment for different types of non-Newtonian fluid to find out the phenomena of the flow.
- ✓ Studying sedimentation effect on the flow pattern for non-Newtonian fluids in detail is recommended. especially the transient behaviour of the flow should be handled since there may be differences in small intervals of the time.
- ✓ The distance of the channel in experimental work was not long enough to show the stability character of the fluids after passing the obstacle area, hence channel with longer distances is recommended to obtain more accurate results. for both Newtonian and Non-Newtonian fluid.

REFERENCES

- [1] Moglen, G.E. (2015). Fundamentals of open channel flow, CRC Press, Boca Raton.
- [2] Deng, J., Yin, H., Kong, F., Chen, J., Dronova, I. and Pu, Y. (2020). Determination of runoff response to variation in overland flow area by flow routes using UAV imagery. *Journal of Environmental Management*, vol. 265, pp.109868.
- [3] Cavalcante, H., Araújo, F., Becker, V. and Barbosa, J.E.D.L. (2021). Internal phosphorus loading potential of a semiarid reservoir: an experimental study. *Acta Limnologica Brasiliensia*, vol.33, pp.e6.
- [4] Ding, J., Liu, J., Chang, X.B., Zhu, D. and Lassen, S.B., (2020). Exposure of CuO nanoparticles and their metal counterpart leads to change in the gut microbiota and resistome of collembolans. *Chemosphere*, vol. 258, pp.127347
- [5] Alam, M.S. and Khan, M.A.H., 2014. MHD effects on mixed convection flow through a diverging channel with circular obstacle. *Procedia Engineering*, 90, pp.403-410
- [6] Chanson, H. (2002). *Hydraulics of stepped chutes and spillways*. CRC Press. Boca Raton.
- [7] Subramanya, K. (2009). *Flow in open channels*. In *Vascular* (Issue January 2010).
- [8] Chalgeri, V.S., Jeong, J.H., (2019). Flow regime identification and classification based on void fraction and differential pressure of vertical two-phase flow in rectangular channel. *International Journal of Heat and Mass Transfer*, vol. 132, pp. 802-816.
- [9] Sturm, T.W., (2001). *Open channel hydraulics*, McGraw-Hill, New York.
- [10] Chaudhry, M.H., (2008). *Open-channel flow*. Springer, Switzerland.
- [11] Cant, S., (2001). Book review of “S. B. Pope, *Turbulent Flows*, Cambridge University Press, Cambridge, U.K., 2000, 771 pp.” *Combustion and Flame*, vol. 125(4), pp.1361-1362.
- [12] Zhou, J. (2017). *Physics of environmental flows interacting with obstacles*. Department of Civil and Environmental Engineering, Doctor of, pp.167.
- [13] Natasha, G., Suharjito, & Noviantri, V. (2019). Saint-venant model analysis of trapezoidal open channel water flow using finite difference method. *Procedia Computer Science*, vol.157,pp. 6–15.
- [14] Gond, L., Perret, G., Mignot, E., & Riviere, N. (2019). Analytical prediction of the hydraulic jump detachment length in front of mounted obstacles in supercritical open-channel flows Analytical prediction of the hydraulic jump detachment length in front of mounted obstacles in supercritical open-channel flows. *Physics of Fluids*, vol. 31, pp. 1–11.
- [15] Ahmed, S., Hossain, A., Hossain, Z., & Molla, M. (2023). Forced convection of non-Newtonian nanofluid in a sinusoidal wavy channel with response surface analysis and sensitivity test. *Results in Engineering*, vol. 19(June), pp. 101360.
- [16] Tsynaeva, A. and Nikitin, M., (2016). Numerical modeling of rectangular channel with shallow dumbbell dimples-based Code Saturne. *Proceedings of the Institute for System Programming of the RAS (Proceedings of ISP RAS)*, vol. 28(1), pp.185-196.
- [17] Silva, G. and Ginzburg, I., (2015). The permeability and quality of velocity field in a square array of solid and permeable cylindrical obstacles with the TRT–LBM and FEM Brinkman schemes. *Comptes Rendus. Mécanique*, vol. 343(10-11), pp.545-558.
- [18] Young, T.J. and Vafai, K., (2016). Experimental and Numerical Investigation of Forced Convective Characteristics of Arrays of Channel Mounted Obstacles. *Journal of Heat Transfer*, vol. 121 (34).

- [19] Gustavo, V. (2011). Newtonian and Non-Newtonian Flow. Food Engineering, II, pp. 1–10.
- [20] Majeed, J. G. (2016). Chapter 3 Non-Newtonian fluid.
- [21] Nouraei, S., Anvari, A., Abed, A. M., Ali Akbari, O., Montazerifar, F., Baghaei, S., & Fatholahi, M. (2023). Heat transfer and entropy analysis for nanofluid flow in a semi-circular open cavity under mixed convection. Alexandria Engineering Journal, vol. 64, pp. 309–334.
- [22] Jobson, H.E. and Froehlich, D.C., (1988). Basic hydraulic principles of open-channel flow. US Geological Survey. pp. 88-707.
- [23] Sheynoy, A. (2018). Introduction: Non-Newtonian Fluids. In Heat Transfer to Non-Newtonian Fluids: Fundamentals and Analytical Expressions (First Edit, pp. 1–10).
- [24] Chanson, H. (1982). Hydraulics of open channels, Elsevier's Science and Technology. UK
- [25] Habib, R., Yadollahi, B., Karimi, N., & Hossein, M. (2020). On the unsteady forced convection in porous media subject to inlet flow disturbances-A pore-scale analysis. International Communications in Heat and Mass Transfer, vol, 116, pp. 104639.
- [26] Sharma, J., Ahammad, N. A., Wakif, A., Ali, N., Dong, J., & Weera, W. (2023). Solutal effects on thermal sensitivity of casson nanofluids with comparative investigations on Newtonian (water) and non-Newtonian (blood) base liquids. Alexandria Engineering Journal, vol. 71, pp. 387–400.
- [27] Greco, M., Mirauda, D., & Volpe Plantamura, A. (2014). Manning's roughness through the entropy parameter for steady open channel flows in low submergence. Procedia Engineering, vol. 70, pp. 773–780.
- [28] Wang, C.T., Chen, Y.M., Qi, Z.Q., Wang, Y.T. and Yang, Y.C., (2014). Types of simplified flow channels without flow obstacles in microbial fuel cells. International journal of hydrogen energy, vol. 39(26), pp. 14306-14311.
- [29] Pimenta, F., & Alves, M. A. (2023). Steady-state simulation of the film-casting process for Newtonian and viscoelastic fluids. Journal of Non-Newtonian Fluid Mechanics, vol. 322, pp. 105132.
- [30] Binder, B. J. (2019). Steady two-dimensional free-surface flow past disturbances in an open channel: Solutions of the Korteweg-de Vries equation and analysis of the weakly nonlinear phase space. Fluids, vol. 4(1), pp. 1–23.
- [31] Jebelli, A., Mahabadi, A., Saeid Zare, M., & Ahmad, R. (2022). Numerical simulations of lateral input effect in an open channel to reduce disturbances in the mainstream channel using CFD. Water-Energy Nexus, vol. 5, pp. 39–49.
- [32] Cahoon, J., & Hoshino, T. (2003). Flume for Teaching Spatially Varied Open-Channel Flow. Journal of Hydraulic Engineering, vol. 129(10), pp. 813–816.
- [33] Chen, Y. C., & Yang, H. C. (2023). Movement characteristics of the inclined surface flow of the open channel on the nanoscale surface. Heliyon, vol. 9(7), pp. e17677.
- [34] Esmaeli, P., Boudaghpour, S., Rostami, M., & Mirzaee, M. (2022). Experimental investigation of permeability and length of a series of spur dikes effects on the control of bank erosion in the meandering channel. Ain Shams Engineering Journal, vol. 13(4), pp. 101701.
- [35] Zayed, M., El Molla, A., & Sallah, M. (2018). An experimental study on angled trash screen in open channels. Alexandria Engineering Journal, vol. 57(4), pp. 3067–3074.
- [36] Fenton, J. D. (2008). Obstacles in streams and their roles as hydraulic structures. Pisa, Italy, pp. 15–22.

- [37] Mean, S., Unami, K., Okamoto, H., & Fujihara, M. (2022). A thorough description of one-dimensional steady open channel flows using the notion of viscosity solution. *Applied Mathematics and Computation*, vol. 415, pp.126730.
- [38] Qasim, R. M., Abdulhussein, I. A., Mohammed, A. A., & Maatooq, Q. A. (2020). The effect of the obstacle on the hydraulic response of the composite hydraulic structure, vol. 12(3), pp.159–172.
- [39] Guenther, P., Felder, S. and Chanson, H. (2013). Flow aeration, cavity processes and energy dissipation on flat and pooled stepped spillways for embankments. *Environmental Fluid Mechanics*, vol. 13(5), pp.503–525.
- [40] Aal, G.M.A., Sobeah, M., Helal, E. and El-Fooly, M. (2018). Improving energy dissipation on stepped spillways using breakers. *Ain Shams Engineering Journal*, vol. 9(4), pp.1887–1896.
- [41] Felder, S. and Chanson, H. (2014). Effects of step pool porosity upon flow aeration and energy dissipation on pooled stepped spillways. *Journal of Hydraulic Engineering*, vol. 140(4), pp. 04014002.
- [42] Al-Husseini, T. R. (2016). A Novel experimental work and study on flow and energy dissipation over stepped spillways. *Journal of Babylon University-Engineering Sciences*, vol. 24(4), pp.1050–1063.
- [43] Gonzalez, C.A., Takahashi, M. and Chanson, H. (2008). An experimental study of effects of step roughness in skimming flows on stepped chutes. *Journal of Hydraulic Research*, vol. 46, pp. 24–35.
- [44] Kordnaeij, M., Asghari Pari, S.A., Sajjadi, S.M. and Shafai Bajestan, M. (2017). Experimentally Comparisons of the effect of porous sheets and porous obstacles in controlling turbidity current. *Water and Soil Science*, vol. 27(1), pp. 43–54.
- [45] Jasim, R. A., Hussen, W. Q., Abdullah, M. F., & Zulkifli, R. (2023). Numerical Simulation of Characterization of Hydraulic Jump Over an Obstacle in an Open Channel Flow. vol. 1(1), pp.1–15.
- [46] Kumar, R., Nadda, R., Kumar, S., Saboor, S., & Saleel, C. A. (2023). Convective heat transfer enhancement using impingement jets in channels and tubes : A comprehensive review. *Alexandria Engineering Journal*, vol. 70, pp.349–376.
- [47] Abbas, A. M., Mohammed, A. Y. (2020). Roughness Effect on Velocity Distribution in Selected Reach of Shatt al-Arab River. *Journal of Engineering*, vol. 26(8), pp.46-58 .
- [48] Bora, U. M. (2018). An Experimental Study on Effect of Flexibility of Vegetation on Resistance to Flow. *International Research Journal of Engineering and Technology (IRJET)*, vol. 5(2), pp.1–15.
- [49] Thappeta S. K., Bhallamudi, S. M., Fiener, F., Narasimhan, B., (2017) “Resistance in Steep Open Channels Due to Randomly Distributed Macro Roughness Elements at Large Froude Numbers,” *Journal of Hydraulic Engineering*, vol.22, pp. 12,2017.
- [50] Mulahasan, S.,(2016) “Hydrodynamics of Large Scale Roughness in Open Channels,” Ph.D. thesis, Division of Civil Engineering, Cardiff School of Engineering, Cardiff University.
- [51] Wang, X., Ye, C., Wang, B., (2015). Experimental Study on Velocity Profiles with Different Roughness Elements in a Flume. *Acta Geophysica*, vol. 63(6), pp.1685–1705.
- [52] Na’inna, A.M., Phylaktou, H.N. and Andrews, G.E., (2014). Effects of obstacle separation distance on gas explosions: the influence of obstacle blockage ratio. *Procedia Engineering*, vol. 84, pp.306-319.
- [53] Ciccarelli, G., Fowler, C.J. and Bardon, M.J.S.W., (2005). Effect of obstacle size and spacing on the initial stage of flame acceleration in a rough tube. *Shock waves*, vol.14, pp.161-166.

- [54] Gamezo, V. N., & Oran, E. S. (2007). Origins of the deflagration-to-detonation transition in gas-phase combustion. *Combust. Flame*, vol. 148, pp. 4–47.
- [55] Chaulagain, D., Rimal, P.R., Ngando, S.N., Nsafon, B.E.K., Suh, D. and Huh, J.S., (2023). Flood susceptibility mapping of Kathmandu metropolitan city using GIS-based multi-criteria decision analysis. *Ecological Indicators*, vol. 154, pp.110653.
- [56] Ghobadian, R., Basiri, M., & Tabar, Z. S. (2018). Interaction between channel junction and bridge pier on flow characteristics. *Alexandria Engineering Journal*, vol. 57(4), pp. 2787–2795.
- [57] Ikani, N., Pu, J. H., Zang, S., Hamid, E., Al-qadami, H., & Razi, A. (2024). Detailed turbulent structures investigation around piers group induced flow. *Experimental Thermal and Fluid Science*, vol. 152, pp. 111112.
- [58] Liao, B., & Chen, S. (2015). Experimental study of flow past obstacles by PIV. *Procedia Engineering*, vol. 126, pp.537–541.
- [59] Lee, J. H. S., Peraldi, O., & Knystautas, R. (1986). Criteria for transition to detonation in tubes. *Proc. Combust. Inst.*, pp. 1629–1637.
- [60] Masri, A. R., Ibrahim, S. S., & Nehzat, N. et al. (2000). Experimental study of premixed flame propagation over various solid obstructions. *Exp Therm Fluid Sci*, vol. 21(1), pp. 109e16.
- [61] Oh, K. H., Kim, H., Kim, J. B., & Al., E. (2001). A study on the obstacle-induced variation of the gas explosion characteristics. *J Loss Prev Process Ind*, vol.14(6), pp.597e602.
- [62] Ciccarelli, G., Johansen, C. T., & Parravani, M. (2010). The role of shock-flame interactions on flame acceleration in an obstacle laden channel. *Combustion Flame*, vol. 157, pp.2125e36.
- [63] Han, S. X., Yu, M. J., Yang, X. F., & Al., E. (2020). Effects of obstacle position and hydrogen volume fraction on premixed syngas-air flame acceleration. *Int J Hydrogen Energy*, vol. 45, pp. 29518e32.
- [63] Gamezo, V. N., Khokhlov, A. M., & Oran, E. S. (2001). The influence of shock bifurcations on shock-flame interactions and DDT. *Combustion Flame*, vol. 126, pp.1810e26.
- [65] Oran, E., Gamezo, V. and Khokhlov, A., (2002), January. Effects of boundary layers and wakes on shock-flame interactions and DDT. In 40th AIAA Aerospace Sciences Meeting & Exhibit, pp. 776.
- [66] Ashley, M., Donovan, L., & Brian, J. (2019). Numerical investigation of the effect of obstacle shape on deflagration to detonation transition in a hydrogen-air mixture. *Combust Flame*, vol.209, pp. 891e904.
- [67] Saeid, M. H. S., Khadem, J., & Emami, S. (2021). Numerical investigation of the mechanism behind the deflagration to detonation transition inhomogeneous and inhomogeneous mixtures of H₂-air in an obstructed channel. *Int J Hydrogen Energy*, vol. 46, pp. 21657e71.
- [68] Riviere, N., Vouaillat, G., Launay, G., & Mignot, E. (2017). Emerging Obstacles in Supercritical Open-Channel Flows : Detached Hydraulic Jump versus Wall-Jet-Like Bow Wave. *Journal of Hydraulic Engineering*, vol. 18(2), pp. 1–9.
- [69] Christodoulou, B. G. C. (2013). Equivalent Roughness Of Submerged Obstacles In Open Channel Flows. *Journal of Hydraulic Engineering*, pp. 1.
- [70] Liao, L., Li, Z., Lang, T. and Chen, G., (2015). UV LED array based NLOS UV turbulence channel modeling and experimental verification. *Optics express*, vol. 23(17), pp.21825-21835.
- [71] Higham, J. E., & Brevis, W. (2018). Modification of the modal characteristics of a square cylinder wake obstructed by a multi-scale array of obstacles. *Experimental Thermal and Fluid Science*, vol. 90(August 2017), pp. 212–219.

- [72] Sharma, J., Ahammad, N. A., Wakif, A., Ali, N., Dong, J., & Weera, W. (2023). Solutal effects on thermal sensitivity of cassin nanofluids with comparative investigations on Newtonian (water) and non-Newtonian (blood) base liquids. *Alexandria Engineering Journal*, vol. 71, pp. 387–400.
- [73] Cha, H., Amiri, H. A., Moshafi, S., Karimi, A., Nikkhah, A., Chen, X., Ta, H. T., Nguyen, N. T., & Zhang, J. (2023). Effects of obstacles on inertial focusing and separation in sinusoidal channels: An experimental and numerical study. *Chemical Engineering Science*, vol. 276(April), pp. 118826.
- [74] Wang, L., Si, R. J., Li, R. Z., & Huo, Y. (2018). Experimental investigation of the propagation of deflagration flames in a horizontal underground channel containing obstacles. *Tunn. Undergr. Space Technol.*, vol. 78, pp. 201–214.
- [75] Abdulmajid, N., Phylaktou, M., Herodotos, N., Andrews, & E., G. (2014). Effects of obstacle separation distance on gas explosions: the influence of obstacle blockage ratio. *Procedia Engineering*, vol. 84, pp. 306–319
- [76] Bazin, P. H., Bessette, A., Mignot, E., Paquier, A., & Riviere, N. (2012). Influence of detailed topography when modeling flows in street junction during urban flooding. *J. Disaster Res.*, vol. 7(5), pp.560–566.
- [77] Mignot, E., Zeng, C., Dominguez, G., Li, C. W., Rivière, N., & Bazin, P. H. (2013). Impact of topographic obstacles on the discharge distribution in open-channel bifurcations. *Journal of Hydrology*, vol. 494, pp. 10–19.
- [78] Aiderman, N. J. (2016). Non-Newtonian Fluids: Introduction and Guide To Classification and Characteristics. In *Engineering Sciences Data Unit, Data Items*.
- [79] Maritz, R., & Doungmo Goufo, E. F. (2014). Newtonian and non-newtonian fluids through permeable boundaries. *Mathematical Problems in Engineering*.
- [80] Duan, X., Xiao, J., Zhou, A., Wang, S., & Wen, J. (2023). Numerical investigation on flow and heat transfer performance of non-Newtonian Bingham fluid in novel spiral wound tube heat exchangers. *Case Studies in Thermal Engineering*, vol. 49(July), pp. 103352.
- [81] Poole, R. J. (2023). Inelastic and flow-type parameter models for non-Newtonian fluids. *Journal of Non-Newtonian Fluid Mechanics*, vol. 320, pp. 105106.
- [82] Maroofiazar, R., & Fahimi Farzam, M. (2021). Experimental investigation of energy harvesting from sloshing phenomenon: Comparison of Newtonian and non-Newtonian fluids. *Energy*, pp. 225.
- [83] Malekian, S., Ahmad Louyd Arab, M., & Najjar, R. (2021). Effects of zero-shear rate viscosity and interfacial tension on immiscible Newtonian-Non-Newtonian fluids morphology in radial displacement inside the Hele-Shaw cell. *Journal of the Taiwan Institute of Chemical Engineers*, vol. 127, pp. 46–55.
- [84] Amaratunga, M., Raben jaf imanantsoa, H.A. and Time, R.W., (2019). Comparison of oscillatory flow conditions in Newtonian and non-Newtonian fluids using PIV and high-speed image analysis. *Flow Measurement and Instrumentation*, vol. 70, pp.101628.
- [85] Fazli, M., Rudman, M., Kuang, S., & Chryss, A. (2023). Application of immersed boundary methods to non-Newtonian yield-pseudoplastic flows. *Applied Mathematical Modelling*, vol. 124, pp. 532–552.
- [86] Soligo, G. and Rosti, M.E., (2023). Non-Newtonian turbulent jets at low-Reynolds number. *International Journal of Multiphase Flow*, vol. 167, pp.104546.

- [87] Mustafa, A., Eser, A., Aksu, A. C., Kiraz, A., Tanyeri, M., Erten, A., & Yalcin, O. (2020). A micropillar-based microfluidic viscometer for Newtonian and non-Newtonian fluids. *Analytica Chimica Acta*, vol. 1135, pp. 107–115.
- [88] Hasnain, M., Nadeem, S., Ullah, A., Ayesb, S., Ahammad, N. A., & Eldin, S. M. (2024). On the steady flow of non-newtonian fluid through multi-stenosed elliptical artery : A theoretical model p w p w. *Ain Shams Engineering Journal*, vol. 15(1), pp. 102262.
- [89] Romano, M. G., Alberini, F., Liu, L., J. H. Simmons, M., Stitt, E. H., Simmons, M. J. H., & Stitt, E. H. (2023). Comparison between RANS and 3D-PTV measurements of Newtonian and non-Newtonian fluid flows in a stirred vessel in the transitional regime. *Chemical Engineering Science*, vol. 267, pp. 118294.
- [90] Jebelli, A., Zare, M. S., Lotfi, N., & Yagoub, M. C. E. (2023). Numerical simulation and optimization of a circular open channel for fish farming using Computational Fluid Dynamics (CFD). *Water-Energy Nexus*, vol. 6, pp. 96–111.
- [91] Keramaris, E., Pechlivanidis, G., Kasiteropoulou, D., Michalolias, N., & Liakopoulos, A. (2016). Experimental and numerical study of turbulent flow in open channels with impermeable and porous bed. *Procedia Engineering*, vol. 162(2016), pp. 381–387.
- [92] Nguyen, V.T. (2015). 3D numerical simulation of free surface flows over hydraulic structures in natural channels and rivers. *Applied Mathematical Modelling*, vol. 39(20), pp. 6285–6306.
- [93] Ben, S., Ben, R., Mahjoub, N., Bournot, H., & Le, G. (2017). Numerical study of sediment transport in turbulent two-phase flows around an obstacle. *Applied Mathematical Modelling*, vol. 45, pp. 97–122.
- [94] Ibrahim, M. M., Ashmawy, A., Dalia, M., & Refaey, M. A. (2023). Numerical investigation of the effect of hot-water outlet inclination angle on the temperature dilution in open channel flow. *Ain Shams Engineering Journal*, vol. 14(12), pp. 102234.
- [95] Natasha, G., Suhajito, & Noviantri, V. (2019). Saint-venant model analysis of trapezoidal open channel water flow using finite difference method. *Procedia Computer Science*, 157, 6–15.
- [96] Edalati-nejad, A., Fanaee, S.A., Ghodrat, M., Salehi, F. and Khadem, J., (2020). The time dependent investigation of methane-air counterflow diffusion flames with detailed kinetic and pollutant effects into a micro/macro open channel. *Case Studies in Thermal Engineering*, vol. 18, pp.100603.
- [97] Mahmood, F. T., Das, A., Smriti, R. B., Hakim, M. A., Saha, S., & Hasan, M. N. (2023). Role of wall-mounted flexible flow modulator on thermo-hydraulic characteristics of pulsating channel flow. *Results in Engineering*, pp. 100941.
- [98] Crnković, B., Črnjarić-Žic, N., & Kranjčević, L. (2009). Improvements of semi-implicit schemes for hyperbolic balance laws applied on open channel flow equations. *Computers and Mathematics with Applications*, vol. 58(2), pp. 292–309.
- [99] Mean, S., Unami, K., Okamoto, H. and Fujihara, M., (2022). A thorough description of one-dimensional steady open channel flows using the notion of viscosity solution. *Applied mathematics and computation*, vol. 415, pp.126730.
- [100] Zhang, P., Duan, Y., Zhong, Q., Li, D., Yang, S., Hu, J., & Li, W. (2022). Self-similarity of spanwise rotational motions' population trends in decelerating open-channel flow. *Theoretical and Applied Mechanics Letters*, vol. 12(3), pp. 4–10.
- [101] Asif, M., Ghani, U., Pasha, G. A., Ullah, M. K., & Ali, S. (2023). 3D numerical investigation of flow behavior in an open channel with uniform and layered vegetation patches under varying submergence conditions. *Ain Shams Engineering Journal*, vol. 15(1), pp. 102288.

- [102] Abduo, S. S., Elmoustafa, A. M., & Salam, M. S. A. (2021). Study of counteracting the secondary flow in open channel bends. *Ain Shams Engineering Journal*, vol. 12(3), pp. 2425–2433.
- [103] Majeed, H.Q., Ghazal, A.M. and Al-Hadeethi, B., (2022). Experimental and Numerical Study of Open Channel Flow with T-Section Artificial Bed Roughness. *Mathematical Modelling of Engineering Problems*, vol. 9(6).
- [104] Seyedashraf, O. and Akhtari, A.A., 2017. Three-dimensional CFD study of free-surface flow in a sharply curved 30 open-channel bend. *Journal of Engineering Science and Technology Review*, vol. 10(3), pp.85-89.
- [105] Chara, Z., 2020, November. CFD-DEM simulations in water channel flow. In *AIP Conference Proceedings*, vol. 2293, p. 030018.
- [106] Burman, A., 2016. Open-Channel Flow Over Naturally Rough Surfaces: A CFD Study.



APPENDICES

APPENDIX- 1. Froude number for all water cases (Rectangular obstacle)

S=0°, Q=0.0035 m ³ /s						
X [m]	Exp. Left	Exp. Center	Exp. Right	Num. Center	Num. Left	Num. Right
0.6509	0.30463	0.338151	0.30463	0.306834	0.341828	0.301133
1.4	0.301628	0.335006	0.258539	0.304807	0.350144	0.254957
1.6	0.348148	0.422689	0.348148	0.344798	0.444869	0.324233
1.75	0.395643	0.426734	0.395643	0.393723	0.450563	0.372406
1.9	0.421951	0.473987	0.421951	0.410745	0.488025	0.381356
2.05	0.412853	0.421951	0.412853	0.401866	0.432676	0.369133
2.2	0.474826	0.504598	0.474826	0.441505	0.495576	0.396579
2.35	0.445809	0.453712	0.445809	0.430206	0.464826	0.383538
2.5	0.482109	0.666732	0.482109	0.443377	0.607965	0.375784
2.65	0.572161	0.673909	0.572161	0.543414	0.670036	0.432479
2.75	0.67699	0.756003	0.67699	0.588764	0.725203	0.435206
2.9	0.603493	0.685712	0.603493	0.551757	0.651643	0.406242
3.05	0.67699	0.756003	0.67699	0.610503	0.716032	0.440517
3.4	0.769975	0.844891	0.769975	0.686238	0.775924	0.479702
3.92	0.705964	0.78728	0.705964	0.754942	0.853086	0.463788

S=0°, Q=0.0040 m ³ /s						
0.6509	0.298714	0.335006	0.298714	1.48668	1.542129	1.468421
1.4	0.338151	0.331946	0.338151	1.550669	1.537267	1.539806
1.6	0.430898	0.397819	0.430898	1.738634	1.701826	1.751213
1.75	0.426734	0.418757	0.426734	1.759614	1.758861	1.757605
1.9	0.414933	0.422689	0.414933	1.740977	1.723737	1.724229
2.05	0.532991	0.473987	0.532991	1.94416	1.840223	1.930233
2.2	0.478704	0.543982	0.478704	1.821459	1.939181	1.803763
2.35	0.563127	0.599615	0.563127	1.92042	2.042923	1.914324
2.5	0.580514	0.603082	0.580514	1.817242	1.954791	1.8029
2.65	0.576698	0.689851	0.576698	1.883438	2.051794	1.879393
2.75	0.65714	0.673909	0.65714	1.844713	1.995979	1.832809
2.9	0.742854	0.615445	0.742854	1.975694	1.882473	1.940047
3.05	0.718492	0.59838	0.718492	1.962234	1.834382	1.944015
3.4	0.685712	0.552686	0.685712	1.90524	1.792974	1.907664
3.92	0.663843	0.543144	0.663843	1.923518	1.816878	1.931204

S=0°, Q=0.0045 m³/s						
0.6509	0.293144	0.328985	0.293144	0.296704	0.327248	0.297262
1.4	0.331963	0.320512	0.331963	0.322486	0.319402	0.321493
1.6	0.42271	0.383875	0.42271	0.396442	0.385729	0.409171
1.75	0.418778	0.414953	0.418778	0.410334	0.418604	0.41707
1.9	0.407608	0.414953	0.407608	0.405242	0.403618	0.405762
2.05	0.522248	0.460656	0.522248	0.512205	0.463943	0.510369
2.2	0.469431	0.528388	0.469431	0.45764	0.52656	0.457787
2.35	0.550498	0.587529	0.550498	0.523535	0.596466	0.524494
2.5	0.564185	0.589854	0.564185	0.48797	0.56133	0.487717
2.65	0.563154	0.649504	0.563154	0.546121	0.643035	0.541293
2.75	0.63562	0.885733	0.63562	0.555434	0.679762	0.601232
2.9	0.718527	0.674219	0.718527	0.65211	0.6201	0.642325
3.05	0.696413	0.663875	0.696413	0.652425	0.591782	0.658413
3.4	0.663255	0.615475	0.663255	0.630892	0.576292	0.653551
3.92	0.639726	0.577509	0.639726	0.729192	0.669105	0.754575

S=1.0156°, Q=0.0035 m³/s						
0.6509	0.894515	1.194856	0.894515	1.072447	1.247669	1.074037
1.4	0.505893	0.59838	0.505893	0.542637	0.651877	0.544011
1.6	0.422192	0.664757	0.369418	0.429951	0.691727	0.372778
1.75	0.480582	0.7055	0.432524	0.523563	0.722014	0.448852
1.9	0.428541	0.720873	0.482109	0.427775	0.74338	0.459304
2.05	0.486552	0.656981	0.486552	0.506969	0.695245	0.487439
2.2	0.533731	0.729828	0.533731	0.489248	0.746412	0.476818
2.35	0.399353	0.624756	0.399353	0.425268	0.657651	0.406996
2.5	0.604355	0.59903	0.604355	0.491456	0.63535	0.473358
2.65	0.663843	0.65766	0.663843	0.672628	0.691715	0.553121
2.75	0.69095	1.044871	0.69095	0.678624	1.058393	0.777912
2.9	0.895604	1.100942	0.895604	0.947916	1.199225	0.888311
3.05	1.032897	1.275861	1.032897	0.987103	1.404816	0.998373
3.4	1.044871	1.233635	1.044871	1.116043	1.314756	1.148053
3.92	1.191804	1.450897	1.112351	1.249986	1.515854	1.181165

S=1.0156°, Q=0.0040 m³/s						
0.6509	1.007258	1.315877	0.940108	1.147061	1.228061	1.118105
1.4	0.556482	0.624756	0.303536	0.527723	0.658141	0.368424
1.6	0.474966	0.748788	0.369418	0.4602	0.68099	0.393498
1.75	0.432524	0.664757	0.432524	0.468287	0.658993	0.448922
1.9	0.508892	0.720873	0.508892	0.495903	0.693378	0.492206
2.05	0.474826	0.589826	0.474826	0.500773	0.617989	0.491129
2.2	0.543982	0.696844	0.543982	0.52451	0.661587	0.51969
2.35	0.535207	0.610054	0.535207	0.537457	0.605934	0.523607

2.5	0.61928	0.739126	0.756897	0.50771	0.701994	0.564158
2.65	0.742854	0.672815	0.742854	0.75797	0.732378	0.647714
2.75	0.710655	0.917753	0.710655	0.717168	0.957447	0.877147
2.9	1.087963	1.049707	1.087963	1.124427	1.110299	1.046268
3.05	1.112351	1.067831	1.152077	1.031488	1.20312	1.083293
3.4	1.032133	1.172424	1.032133	1.191606	1.211786	1.223786
3.92	1.350711	1.29299	1.310985	1.375922	1.356594	1.362239

S=1.0156°, Q=0.0045 m³/s						
0.6509	0.756897	0.725309	0.756897	0.838051	0.781253	0.836844
1.4	0.535207	0.583517	0.437897	0.516045	0.62479	0.440981
1.6	0.505893	0.703909	0.505893	0.466579	0.658797	0.457684
1.75	0.440714	0.642216	0.440714	0.472608	0.644996	0.442835
1.9	0.512871	0.635197	0.512871	0.486623	0.637223	0.471888
2.05	0.5103	0.6129	0.5103	0.523634	0.658972	0.50857
2.2	0.52774	0.659405	0.52774	0.493042	0.67481	0.480281
2.35	0.474826	0.554755	0.474826	0.482493	0.593066	0.469121
2.5	0.67699	0.703909	0.738534	0.546665	0.684713	0.565555
2.65	0.685712	0.642216	0.685712	0.688747	0.7244	0.615756
2.75	0.814157	1.100942	0.814157	0.76149	1.043973	0.927378
2.9	1.032133	1.046257	1.032133	1.078228	1.104574	0.990268
3.05	1.194139	1.086288	1.194139	1.077959	1.206473	1.095214
3.4	1.115313	0.970238	1.115313	1.235233	0.981741	1.234617
3.92	1.363609	1.268772	1.363609	1.346686	0.766315	1.351126

S=1.406°, Q=0.0035 m³/s						
0.6509	0.895604	1.032897	0.895604	1.041639	1.120463	1.045384
1.4	0.671506	0.986908	0.671506	0.847254	1.032529	0.85548
1.6	0.393275	0.583862	0.393275	0.373956	0.554475	0.354896
1.75	0.399353	0.653261	0.399353	0.4138	0.61342	0.38724
1.9	0.422445	0.718019	0.422445	0.393412	0.695252	0.379245
2.05	0.474232	0.688498	0.474232	0.497437	0.725974	0.459366
2.2	0.492356	0.748788	0.492356	0.447651	0.761369	0.426053
2.35	0.505893	0.681173	0.505893	0.531409	0.718872	0.486315
2.5	0.87399	0.733544	0.87399	0.596145	0.763136	0.566563
2.65	0.69095	0.67619	0.69095	0.697589	0.731522	0.560703
2.75	0.721673	1.065982	0.721673	0.615972	1.054005	0.837456
2.9	0.956277	1.058946	0.956277	1.163505	1.155851	0.923394
3.05	0.986908	1.18092	0.986908	0.947444	1.499635	1.036165
3.4	1.015432	1.233635	1.015432	1.25199	1.376286	1.233662
3.92	1.280203	1.256272	1.280203	1.461638	1.505619	1.434052

S=1.406°, Q=0.0040 m³/s						
0.6509	1.015432	1.271258	0.942902	1.782601	1.201381	1.087814
1.4	1.000098	1.328747	1.000098	1.037561	1.181803	0.969051
1.6	0.449457	0.769306	0.449457	0.656869	0.651972	0.388306
1.75	0.443475	0.681173	0.443475	0.651099	0.669496	0.415374
1.9	0.592212	0.734433	0.592212	0.712737	0.725534	0.491034
2.05	0.462224	0.656981	0.462224	0.643201	0.702025	0.455811
2.2	0.628569	0.729828	0.628569	0.731533	0.729042	0.539348
2.35	0.549111	0.664757	0.549111	0.685725	0.702457	0.516501
2.5	0.870371	0.729828	0.870371	0.761629	0.754822	0.613017
2.65	0.83414	0.754212	0.83414	0.777681	0.871495	0.683391
2.75	0.738534	1.280203	0.769306	0.726954	0.807685	0.956157
2.9	1.032133	1.268772	1.032133	0.865026	1.443429	1.024498
3.05	1.271258	1.193458	1.271258	0.901869	1.235561	1.236085
3.4	1.16975	1.191804	1.16975	0.909138	1.256615	0.758872
3.92	1.350711	1.233635	1.350711	0.907327	1.38993	0.815855

S=1.406°, Q=0.0045 m³/s						
0.6509	1.809228	1.480361	1.032133	7.874409	1.145825	1.240066
1.4	1.315877	1.41804	1.23089	8.069232	1.148586	1.26548
1.6	0.769306	0.923168	0.512871	16.30982	0.45443	0.619769
1.75	0.662038	0.77376	0.417518	24.64644	0.469956	0.581434
1.9	0.705964	0.729828	0.486552	17.72984	0.515829	0.663998
2.05	0.74761	0.664757	0.504598	23.5086	0.545579	0.634532
2.2	0.78728	0.829029	0.563127	16.78945	0.605957	0.676549
2.35	0.76174	0.664757	0.550471	22.42717	0.597795	0.628579
2.5	0.800079	0.872311	0.807204	12.7739	0.671591	0.673769
2.65	0.777105	0.888318	0.698869	13.14339	0.782174	0.757062
2.75	0.762845	0.47377	1.032897	13.41325	0.739628	0.613446
2.9	0.85714	1.307368	0.962469	9.31711	1.236073	1.173197
3.05	0.947236	1.41804	1.046257	7.509686	1.103183	1.276726
3.4	0.897158	0.756897	1.307821	8.786282	1.333873	0.558109
3.92	0.982984	0.692376	1.41804	6.635702	1.600274	0.52121

CURRICULUM VITAE

Azeez Majeed MOHAMMED

██████████	████████████████████
██████████	██████████
██████████	██████
██████████	████████████████████
██████████	██
██████████	████████████████████

Service	Used	Not used	Don't know
Used a service to help with my mental health	85%	15%	0%
Used a service to help with my physical health	75%	25%	0%
Used a service to help with my social health	65%	35%	0%

[REDACTED]
 [REDACTED]

[REDACTED]	[REDACTED]
[REDACTED]	[REDACTED]
[REDACTED]	[REDACTED]

ACADEMIC ACTIVITIES

Paper:

1. Mohammed, A.M., Kapan, S., Sen, M., Celik, N. Effect of vibration on heat transfer and pressure drop in a heat exchanger with turbulator, *Case Studies in Thermal Engineering*, 2021, 28, 101680.

Proceedings:

1. Mohammed, A.M., Ahmed, T.M., Kıstak C., S., Celik, N. Parametric Study of Rectangular Obstacles Effects on flow in open channels, International Capital Conference on Scientific Research and Technological Developments, April 26-27, 2024/ Amsterdam, Netherlands