

CFD ANALYSIS OF HYDRAULIC JUMP UNDER VARYING CHANNEL CONDITIONS

**A Thesis Submitted in Partial Fulfilment of the
Requirements for the degree of**

DOCTOR OF PHILOSOPHY

by

HARSHIT KUMAR JAYANT

(2K19/PHD/CE/04)

**Under the Supervision of
Dr. BHARAT JHAMNANI**

DTU



Department of Civil Engineering

DELHI TECHNOLOGICAL UNIVERSITY

(Formerly Delhi College of Engineering)

Shahbad Daulatpur, Main Bawana Road, Delhi-110042. India

June, 2025

©DELHI TECHNOLOGICAL UNIVERSITY, DELHI-2025
ALL RIGHTS RESERVED

CFD ANALYSIS OF HYDRAULIC JUMP UNDER VARYING CHANNEL CONDITIONS

**A Thesis Submitted in Partial Fulfilment of the
Requirements for the degree of**

DOCTOR OF PHILOSOPHY

by

HARSHIT KUMAR JAYANT

(2K19/PHD/CE/04)

**Under the Supervision of
Dr. BHARAT JHAMNANI**

DTU



Department of Civil Engineering

DELHI TECHNOLOGICAL UNIVERSITY

(Formerly Delhi College of Engineering)

Shahbad Daulatpur, Main Bawana Road, Delhi-110042. India

June, 2025

ACKNOWLEDGEMENTS

I would like to take this opportunity to thank all individuals who inspired and assisted me in producing this thesis. It is also my prerogative to thank highly many individuals who have encouraged me as I could not possibly have thought of executing this project. I am pleased to have been surrounded by so much generosity and kind during the course of this though but very rewarding procedure.

I am giving my endless gratitude to my supervisor **Dr. Bharat Jhamnani**, Assistant Professor from Delhi Technological University in the Department of Civil Engineering for his able assistance and guidance in developing this thesis; His constructive advices, stringent criticism and encouragement acted as significant roles in shaping this work. He said some of the most intellectually honest things I've ever been told, and allowed me to develop as an independent scholar. The endless belief he had in me and his constructive criticism has assisted in achieving this thesis.

On this note, I would to give special thanks to **Prof. K. C. Tiwari**, Head of the Department of Civil Engineering at Delhi Technological University for the support given throughout my Ph.D. His openness to embrace new views and avail his direction played a great role in the completion of this study. I would also like to express my appreciation to all the faculties of the Civil Engineering Department for their cooperation and support I received and the contribution that they gave towards my learning's and researches. Their ideas and advices contributed a lot to make my academics so much valuable.

I am very thankful to the distinguished members of my doctoral research committee, **Prof. D. R. Kaushal** (IIT Delhi), **Prof. Muhammad Sharif** (Jamia Millia Islamia, Delhi), **Prof. V. K. Minocha**, **Prof. S. Anbukumar**, **Prof. K.C. Tiwari**, and **Prof. R.M. Singari** (DTU). The feedback they provided me was so reflection-critical that it impacted the research work. Honestly, they dedicated many hours in revising my papers and provided me some valuable advices which I suspect to be the main reason for the diversity of this thesis.

I would like to thank Hydraulic Laboratory and Computer Aided Design (CAD) Laboratory of the Civil Engineering Department of Delhi Technological University. I am especially grateful to the members of the laboratory for their professional advice.

More specifically, I am soaked with gratitude to **Dr. Manvendra Verma, Dr. Rahul Kumar Meena, Dr. Abhishek Prakash Paswan, Dr. Deepak Singh and Dr. Indrajeet Singh**. This encouragement and wise counsel were invaluable in helping to solve the most analytical and difficult questions of my work. They provided guidance in defining what needed to be done, as well as helping to show the way when it was most needed.

I would like to extend my sincere gratitude to my dear friends, **Mr. Prateek Roshan, Mr. Aniket Kumar Sharma, Mr. Goverdhan Meena, Mr. Shivendra Krishna Chaurasia, Mr. Sunil Kumar, Mr. Aman Singh, and Mr. Abhishek Kumar Bharti**. Their unwavering support, camaraderie, and shared commitment to our research work, cultivated during countless discussions at our coffee tables, have been invaluable to this journey. These moments sparked ideas, refined our understanding, and strengthened the resolve to push forward. Their presence has been a source of great motivation and emotional support during difficult times.

I am highly thankful to my wonderful and affectionate parents **Shri. Rajendra Prasad & Smt. Maya Devi** who are fully involved in every phase of life. They are always not only selfless in giving of themselves but are my source of strength and inspiration indeed. I would also greatly and truly appreciate my brother **Mr. Garvit Kumar Jayant** and my sister-in-law **Mrs. Khushboo** for being their supportive selves alongside me in every hardship. I would like especially to thank my cousin for being always beside me and for inspiring me during the toughest times of this work.

I am very much thankful to all of them as they had taken active participation in doing this work and this work would have not been completed without their support.

(Harshit Kumar Jayant)



DELHI TECHNOLOGICAL UNIVERSITY
(Formerly Delhi College of Engineering)
Shahbad Daultpur, Main Bawana Road, Delhi-42

DECLARATION

I hereby certify that the work which is being presented in the thesis entitled **“CFD Analysis of Hydraulic Jump under Varying Channel Conditions”** in partial fulfilment of the requirements for the award of the Degree of Doctor of Philosophy and submitted in the Department of Civil Engineering of the Delhi Technological University, Delhi is an authentic record of my own work carried out during a period from August, 2019 to April, 2024 under the supervision of Dr. Bharat Jhamnani, Assistant Professor, Department of Civil Engineering, Delhi Technological University, Delhi, India.

The matter presented in this thesis has not been submitted by me for the award of any other degree or any other Institution.

Candidate's Signature

This is to certify that the student has incorporated all the corrections suggested by the examiners in the thesis and the statement made by candidate is correct to the best of our knowledge.

Signature of Supervisor

Signature of External Examiner



DELHI TECHNOLOGICAL UNIVERSITY

(Formerly Delhi College of Engineering)
Shahbad Daultapur, Main Bawana Road, Delhi-42

CERTIFICATE BY THE SUPERVISOR

Certified that **Harshit Kumar Jayant** (2K19/PHD/CE/04) has carried out their research work presented in this thesis entitled “**CFD Analysis of Hydraulic Jump under Varying Channel Conditions**” for the award of **Doctor of Philosophy** from Department of Civil Engineering, Delhi Technological University, Delhi, under my supervision. The thesis embodies results of original work, and studies are carried out by the student himself and the contents of the thesis do not form the basis for the award of any other degree to the candidate or to anybody else from this or any other University/Institution.

(Dr. Bharat Jhamnani)

Assistant Professor
Department of Civil Engineering

Date: 03 June, 2025



DELHI TECHNOLOGICAL UNIVERSITY
(Formerly Delhi College of Engineering)

Shahbad Daulatpur, Main Bawana Road, Delhi-42

PLAGIARISM VERIFICATION

Title of the Thesis	CFD Analysis of Hydraulic Jump Under Varying Channel Conditions
Total Pages	121
Name of the Scholar	Harshit Kumar Jayant
Supervisor	Dr. Bharat Jhamnani
Department	Civil Engineering

This is to report that the above thesis was scanned for similarity detection. Process and outcome is given below:

Software used: Turnitin

Similarity Index: 6%

Total Word Count: 25987

Date: 08 June, 2025

Candidate's Signature

Signature of Supervisor

CFD ANALYSIS OF HYDRAULIC JUMP UNDER VARYING CHANNEL CONDITIONS

HARSHIT KUMAR JAYANT

ABSTRACT

Hydraulic jumps are commonly observed phenomena in open channels, including rivers, canals, spillways, and weirs, where they serve a significant function in dissipating excess energy. The observed phenomenon is characterised by the interaction between a high-velocity water flow and a comparatively slower water flow, leading to an abrupt increase in water depth and the subsequent release of a substantial amount of energy. Hydraulic jumps are significant part of various hydraulic processes, as most often they are used not only as energy dissipation tool but also as tool for water aeration and, thus, as chemical mixing agents or as hydraulic jumpers which can lift water levels downstream. Consequently, much research has been conducted with a view of understanding and improving hydraulic jumps.

Hydraulic jump type stilling basin came into knowledge to reduce a large amount of kinetic energy of the flowing fluid in the downstream of the hydraulic structure. This design feature is important in order to exclude the possibility of bed erosion and to achieve the rational use of protective aprons. Within these basins, the water's kinetic energy undergoes a process of transformation, resulting in the generation of turbulence. This turbulence, over time, is dissipated as both heat and sound energy. In order to accomplish this objective, stilling basins are outfitted with various components, including baffle blocks and chutes. Prior research has investigated the behaviour of hydraulic jumps over uneven surfaces in order to improve their characteristics downstream. The present investigation is oriented around examining the influence of different strip macroroughness shapes, their aspect ratios, and arrangements on the characteristics of hydraulic jumps using numerical simulation. This particular area of study has not been extensively explored in previous research activities.

This computational study applied the Computation Fluid Dynamics (CFD), which allowed for an understanding of both types of hydraulic jumps, namely, free and

submerged hydraulic jumps. Key characteristics, such as tail-water depth, sequent depth, jump length, roller length, velocity profiles, flow patterns within the cavity region, Turbulent kinetic energy (TKE), and energy loss are examined over both smooth and macrorough beds. CFD involves predicting fluid flow by solving mathematical equations through simulation. This study takes the advantage of Ansys Fluent software where Reynolds-averaged Navier-Stokes (RANS) equations and standard k - ϵ , re-normalization group (RNG) k - ϵ , realizable k - ϵ , and shear stress transport (SST) k - ω turbulence models are used to predict the mean flow characteristics in turbulent flows. The interface between two immiscible fluids is represented using Volume of Fluid (VOF) model.

To ensure the accuracy of the numerical models, sensitivity analyses of the mesh and turbulence models are performed, and the results are compared with experimental findings for three channel conditions: smooth bed, triangular strip macrorough bed, and trapezoidal strip macrorough bed. Among the various turbulence models RNG k - ϵ model outperforms for predicting both free and submerged hydraulic jumps. Additionally, the standard deviation between the numerically derived results and experimental findings for basic hydraulic jump parameters, such as sequent depth ratio and tailwater depth ratio, is reported below 6%. These findings suggest that Ansys Fluent is a reliable tool for predicting complex phenomena like hydraulic jumps over smooth and rough beds.

To numerically study the hydraulic jump phenomenon, it is crucial to optimize the combination of flow domain and boundary conditions to simulate real-world fluid flow phenomenon cost-effectively, a topic not fully addressed in previous research. In this regard, two approaches are tested for forming hydraulic jumps in a horizontal rectangular smooth channel numerically. The first approach considered the effect of a reservoir just upstream of the sluice gate, while the second approach ignored the reservoir entirely. The results indicated no significant differences in the hydraulic jump's flow behaviour between the two approaches. However, the second approach reduced computational time by up to 50% due to the smaller computational flow domain.

Further investigations were conducted in a prismatic rectangular channel with strip macrorough elements on the downstream bed. Eight distinct macrorough shapes, namely triangular, rectangular, trapezoidal, semicircular, and four new non-regular shapes were compared for their effect on free and submerged hydraulic jump characteristics. It is reported that rough beds generally improve jump characteristics compared to smooth beds. While the shape of the macroroughness has a minor influence on jump characteristics, the triangular strip macrorough shape is the most effective in enhancing hydraulic jump characteristics.

Given the effectiveness of the triangular strip macroroughness, further studies are carried out to explore how the height-to-base width ratio and height-to-wavelength ratio, as well as the arrangement of the strips, affected jump characteristics. The results indicated that varying the height-to-base width ratio does not significantly impact the jump characteristics. However, better energy dissipation is achieved in submerged hydraulic jumps by altering the arrangements of the macrorough elements. Additionally, increasing the spacing between consecutive macrorough elements or decreasing the height-to-wavelength ratio improved the jump characteristics.

Overall, strip type macrorough beds are found effective in enhancing the jump characteristics and a well-designed model can improve the efficiency of hydraulic jump type stilling basins. Further, the numerical simulation technique is well able to predict the complex fluid flow phenomenon and can be beneficial in hydraulic designs.

Keywords: Computational Fluid Dynamics; Energy Dissipation; Hydraulic Jump; Macroroughness

LIST OF PUBLICATIONS

Journals SCI/SCIE

1. Harshit Kumar Jayant, Bharat Jhamnani, Numerical simulation of free and submerged hydraulic jump over trapezoidal and triangular macroroughness, Heliyon, Volume 9, Issue 11, 2023, e22540, ISSN 2405-8440, <https://doi.org/10.1016/j.heliyon.2023.e22540> (SCI-Expanded)
2. Harshit Kumar Jayant, Bharat Jhamnani; Numerical study of submerged hydraulic jumps over triangular macroroughnesses. Journal of Hydroinformatics 1 January 2024; 26 (1): 51–71. <https://doi.org/10.2166/hydro.2023.302>. (SCI-Expanded)

International Conferences

1. CFD Investigation of Hydraulic Jump over Triangular Roughness. Recent advances in Sustainable Construction technologies and Environment-2022. B.S. Abdur Rahman Crescent Institute of Science and Technology.
2. Effect of mesh and turbulence models on predicting the real-world fluid phenomenon using numerical simulation. International Conference on Next Generations Digital Technologies, Akal University, Talwandi Sabo, Bathinda, Punjab.

TABLE OF CONTENTS

ACKNOWLEDGEMENTS.....	ii
DECLARATION	iv
CERTIFICATE BY THE SUPERVISOR.....	v
ABSTRACT	vii
LIST OF PUBLICATIONS	x
TABLE OF CONTENTS.....	xi
LIST OF TABLES.....	xv
LIST OF FIGURES	xvi
LIST OF SYMBOLS, ABBREVIATIONS AND NOMENCLATURE.....	xx
CHAPTER 1: INTRODUCTION.....	1
1.1 General	1
1.2 Hydraulic jump.....	3
1.2.1 Practical applications of hydraulic jump	4
1.2.2 Classical theory of hydraulic jumps.....	4
1.2.3 Types of flows.....	6
1.2.4 Hydraulic jump in a horizontal rectangular channel.....	6
1.2.5 Classification of Jumps.....	8
1.3 Computational Fluid Dynamics.....	10
1.3.2 Turbulence models in computational fluid dynamics (CFD)	11
1.4 Need of the study.....	12
1.5 Objective of the study.....	14
1.6 Outline of thesis.....	14
CHAPTER 2: LITERATURE REVIEW	15
2.1 General	15

CHAPTER 3: METHODOLOGY	42
3.1 General	42
3.2 Physical experimental model.....	43
3.3 Numerical Simulation.....	45
3.3.1 Governing Flow Equations	46
3.3.2 Navier Stokes Equation	46
3.3.3 Volume of Fluid (VOF) Model	47
3.4 Geometry	54
3.5 Meshing.....	55
3.5.1 Mesh sensitivity analysis	56
3.6 Boundary conditions.....	58
3.6.1 Inlet and outlet	58
3.6.2 Near wall treatment.....	58
3.7 Discretization scheme.....	59
3.8 Stability & convergence criteria	59
3.9 Validation of numerical model	60
3.9.1 Longitudinal velocity variation.....	60
CHAPTER 4: RESULT & DISCUSSION	68
4.1 General	68
4.2 Effect of various flume configurations on characteristics of classical hydraulic jump.....	68
4.2.1 CFD generated flow profile including longitudinal velocity profile of hydraulic jump	72
4.3 Comparative analysis of eight (08) different types of strip macroroughness configurations	74
4.3.1 Flow profiles & Velocity profile.....	80

4.3.2 Velocity vector variation	94
4.3.3 Turbulent Kinetic Energy (TKE)	101
4.3.4 Dimensionless Parameters	108
4.3.4.1 Sequent depth ratio (y_2/y_1)	108
4.3.4.2 Tail water depth ratio (y_4/y_1)	109
4.3.4.3 Roller length of jump ratio (L_r/y_1)	109
4.3.4.5 Loss of energy through hydraulic jumps	111
4.4 H/B i.e., height to base width ratio of macroroughness	113
4.4.1 Flow profiles and velocity variation in case of free hydraulic jump	113
4.4.2 Flow profiles and velocity variation in case of submerged hydraulic jump	115
4.5 Possible arrangements with H/B ratio	117
4.5.1 Flow profiles and velocity variation in case of free hydraulic jump	117
4.5.2 Flow profiles and velocity variation in case of submerged hydraulic jump	120
4.6 H/T i.e., height to wavelength ratio of macroroughness	124
4.6.1 Flow profiles and velocity variation in case of free hydraulic jump	124
4.6.2 Flow profiles and velocity variation in case of submerged hydraulic jump	128
4.7 Practical challenges in implementing engineered bed features for hydraulic jump control in real-world systems	132
4.8 Engineering strategies for bed roughness design in energy dissipating structures	133
CHAPTER 5: CONCLUSION, FUTURE SCOPE AND SOCIAL IMPACT ..	134
REFERENCES	137
LIST OF PUBLICATIONS	147
PLAGIARISM REPORT	154

BRIEF CURRICULUM VITAE	156
-------------------------------------	------------

LIST OF TABLES

Table 3.1 Parametric variations of flow for present study (numerical and experimental)	45
--	----

LIST OF FIGURES

Figure 1.1 Stilling basin downstream of the dam, Jhon Martin, United States.....	2
Figure 1.2 Phenomenon of hydraulic jump occurring downstream of the Hira Kund dam, Odisha, India.....	3
Figure 1.3 Definition sketch of specific energy, hydraulic jump and specific force curve [6]	5
Figure 1.4 Definition sketch of hydraulic jump showing relevant parameters	5
Figure 1.5 Sketch showing controlled volume of fluid.....	6
Figure 1.6 Classification of jump based on Froude Number [21].....	9
Figure 1.7 Sketch showing different types of hydraulic jump based on depth	9
Figure 3.1 Definition sketch of free and submerged hydraulic jump showing relevant parameters.....	42
Figure 3.2 (a) Definition sketch of experimental setup; (b) Definition sketch of strip macroroughness; (c) Physical strip macroroughness elements	44
Figure 3.3 Flow chart of adopted methodology for numerical simulation	45
Figure 3.4 Model geometry of simulation setup showing relevant parameters associated with the flume (a) 2-dimensional; (b) 3-dimensional	55
Figure 3.5 Mesh structure allocated to model geometry.....	56
Figure 3.6 Mesh and turbulence model sensitivity analysis on prediction	57
Figure 3.7 Sensitivity analysis of mesh size for horizontal component of velocity profile	57
Figure 3.8 Typical longitudinal velocity profile for a hydraulic jump.....	60
Figure 3.9 Pictorial representation of velocity profile in free hydraulic jump (a) smooth bed (b) trapezoidal macroroughness (c) triangular macroroughness ($Fr_1=5.45$)	62
Figure 3.10 Pictorial representation of velocity profile in submerged hydraulic jump (a) smooth bed (b) trapezoidal macroroughness (c) triangular macroroughness ($Fr_1=6.12$)	63
Figure 3.11 Combined dimensionless flow velocity distribution of submerged hydraulic jump for smooth and macroroughness arrangements at $Fr_1=6.163$	

Figure 3.12 Comparison of a numerical model with experimental model for basic parameters of hydraulic jump (a) y_2/y_1 (b) y_4/y_1	65
Figure 4.1 Sketch of different types of flumes used in first approach	70
Figure 4.2 Sketch of different types of flumes used in second approach	71
Figure 4.3 Numerically simulated flow profile & longitudinal velocity profile at different sections of flume for first approach	72
Figure 4.4 Numerically simulated flow profile & longitudinal velocity profile at different sections of flume for second approach	73
Figure 4.5 Model sketch of adopted different strip macroroughness shapes	79
Figure 4.6 (a-h) Numerically generated flow profiles showing longitudinal velocities at different sections for each model for free hydraulic jump.	82
Figure 4.7 (a-h) Graphical representation of longitudinal velocity profiles of free hydraulic jump	86
Figure 4.8 (a-h) Numerically generated flow profiles showing longitudinal velocities at different sections for each model for submerged hydraulic jump	89
Figure 4.9 Graphical representation of longitudinal velocity profiles of submerged hydraulic jump	93
Figure 4.10 Numerically generated velocity vector for each case of free hydraulic jump	97
Figure 4.11 (a-h) Numerically generated velocity vector for each case of submerged hydraulic jump	100
Figure 4.12 (a-h) Numerically generated turbulent kinetic energy contours for each case of free hydraulic jump	104
Figure 4.13 (a-h) Numerically generated turbulent kinetic energy contours for each case of submerged hydraulic jump	107
Figure 4.14 Sequent depth ratio vs initial Froude number in case of free hydraulic jump	108
Figure 4.15 Tail water depth ratio versus initial Froude number in case of submerged hydraulic jump	109
Figure 4.16 length of jump ratio versus initial Froude number in case of free hydraulic jump	110

Figure 4.17 length of jump ratio versus initial Froude number in case of submerged hydraulic jump	110
Figure 4.18 Relative energy loss versus initial Froude number in case of free hydraulic jump	112
Figure 4.19 Relative energy loss in case of submerged hydraulic jump versus initial Froude number.....	112
Figure 4.20 (a-b) Numerically generated flow profile of free hydraulic jump for H/B i.e., height to base width ratio of macroroughness	113
Figure 4.21 Graphical representation of length of jump ratio versus initial Froude number of free hydraulic jump for H/B i.e., height to base width ratio of macroroughness	114
Figure 4.22 (a-b) Numerically generated flow profile with longitudinal velocity variation of submerged hydraulic jump for H/B i.e., height to base width ratio of macroroughness	115
Figure 4.23 Graphical representation of length of jump ratio versus initial Froude number of submerged hydraulic jump for different possible arrangements of H/B i.e., height to base width ratio of macroroughness	116
Figure 4.24 (a-d) Numerically generated flow profile with longitudinal velocity variation of free hydraulic jump for different possible arrangements of H/B i.e., height to base width ratio of macroroughness	117
Figure 4.25 (a-d) Graphical representation of longitudinal velocity variation of free hydraulic jump for different possible arrangements of H/B i.e., height to base width ratio of macroroughness	119
Figure 4.26 (a-d) Numerically generated flow profile with longitudinal velocity variation of submerged hydraulic jump for different possible arrangements of H/B i.e., height to base width ratio of macroroughness	120
Figure 4.27 (a-d) Graphical representation of longitudinal velocity variation of submerged hydraulic jump for different possible arrangements of H/B i.e., height to base width ratio of macroroughness	122
Figure 4.28 (a-d) Numerically generated flow profile with longitudinal velocity variation of free hydraulic jump for H/I i.e., height to wavelength ratio of macroroughness	124

Figure 4.29 Graphical representation of length of jump ratio versus initial Froude number of free hydraulic jump for H/I i.e., height to wavelength ratio of macroroughness	125
Figure 4.30 (a-d) Graphical representation of longitudinal velocity variation of free hydraulic jump for H/I i.e., height to wavelength ratio of macroroughness	127
Figure 4.31 (a-d) Numerically generated flow profile with longitudinal velocity variation of submerged hydraulic jump for H/I i.e., height to wavelength ratio of macroroughness	128
Figure 4.32 Graphical representation of length of jump ratio versus initial Froude number of submerged hydraulic jump for H/I i.e., height to wavelength ratio of macroroughness	129
Figure 4.33 (a-d) Graphical representation of longitudinal velocity profile of submerged hydraulic jump for H/I i.e., height to wavelength ratio of macroroughness	131

LIST OF SYMBOLS, ABBREVIATIONS AND NOMENCLATURE

CFD	computation fluid dynamics
RANS	Reynolds- averaged Navier-Stokes
RNG	re-normalization group
SST	shear stress transport
VOF	volume of fluid
TKE	turbulent kinetic energy
k	turbulent kinetic energy
ε	dissipation rate of turbulent kinetic energy
ω	specific dissipation rate of turbulent kinetic energy
μ	dynamic viscosity of the fluid
ν	kinematic viscosity
g	gravitational acceleration
u	time-averaged longitudinal velocity at any section
U	the maximum longitudinal velocity in the horizontal direction at any section
Q	discharge
ρ	fluid density
y_0	opening of sluice gate
y_1	initial depth of water in case of hydraulic jump
y_2	tail-water depth of water in case of a free hydraulic jump
y_3	submerged depth of water in case of a submerged hydraulic jump
y_4	tail-water depth of water in case of a submerged hydraulic jump
y_2/y_1	sequent depth ration in case of free hydraulic jump
y_4/y_1	tail-water depth ratio in case of submerged hydraulic jump
L_r/y_1	relative roller length
E_{y_2/y_1}	relative error in prediction of sequent depth ratio
H	height of macroroughness elements
B	the base width of macroroughness elements
Fr_1	Froude number of the inlet section of the hydraulic jump

Re	Reynolds number
C_r	Courant number
L_r	length of roller
L_{j1}	length of hydraulic jump in case of free hydraulic jump
L_{j2}	length of hydraulic jump in case of submerged hydraulic jump
E_1	specific energy beginning of the jump
E_2	specific energy after the jump
R^2	coefficient of correlation
u_i	velocity vector in three-dimensional space, where i represents the cartesian coordinates (x, y, z).
x_i	position vector in three-dimensional space, where i represents the cartesian coordinates (x, y, z).
t	time variable
p	pressure of the fluid
ρ	density of the fluid
x^*	mesh element size
y^+	dimensionless wall distance
t_{ji}	viscous stress tensor
x_i	co-ordinate in the i -axis of a cartesian reference system
μ_t	turbulent eddy dynamic viscosity.
G_k	production term for turbulent kinetic energy
G_b	buoyancy effect term
Y_M	dilatation oscillation effect term
S_k	source terms for turbulent kinetic energy
S_ε	source terms for dissipation rate of turbulent kinetic energy

CHAPTER 1: INTRODUCTION

1.1 General

Hydraulic structures can experience failure for many reasons, and comprehending these factors is of utmost importance to ensure the efficient design and maintenance of such structures (Novak et al. 2017) [1]. The report of Federal Emergency Management Agency (FEMA) published in 2017 [2], listed several common causes for the failure of hydraulic structures. Among these causes, one notable factor is the development of piping and internal soil erosion in embankment dams. It should be noted that erosion and scour are of primary concern for hydraulic engineers when designing hydraulic structures. When water is released from the upstream of a hydraulic structure to the downstream, the potential head of water is converted into kinetic energy. This increase in kinetic energy can result in erosion of the downstream bed and potentially lead to piping underneath the hydraulic structure. Ultimately, this can cause the failure of the hydraulic structure. To address the abovementioned issues, hydraulic engineers apply a technique wherein they strategically design the stilling basins located downstream of the hydraulic structure. This is achieved through the implementation of energy dissipator structures.

Figure 1.1 depicts a stilling basin located downstream of the dam. This structure, built in Jhon Martin, United States, is positioned at the foundation of the spillway to effectively minimize the excess kinetic energy of flowing water by creating a hydraulic jump within the basin. The flow passing over the crest of the spillway undergoes a transition from supercritical to subcritical flow when it reaches the dam toe and meets the normal flow in the downstream side. This transition leads to the hydraulic jump. It is important that the design of the stilling basin promotes the occurrence of a hydraulic jump in its reach so as not to damage the channel located behind the basin. To accomplish this, it is crucial that the post jump depth obtained from the sequent depth relation precisely matches the tail water depth. A stilling basin is composed of a concrete apron and several additional structures, including an end sill, chute blocks, and baffle blocks. Peterka (1958) [3] laid the groundwork and offered valuable insights into the mechanics and principles behind hydraulic jumps in stilling basins. In addition, Bureau of Indian Standards Code 4997 (1968) [4] offers valuable guidance on the

design of hydraulic jump stilling basins. These basins play a crucial role in safely dissipating the energy of flowing water, commonly located downstream of dams, weirs, and spillways. This standard applies to rectangular stilling basins and suggests conducting model testing for designs with falls exceeding 15 m or discharge intensities surpassing $30 \text{ m}^3/\text{s}$.

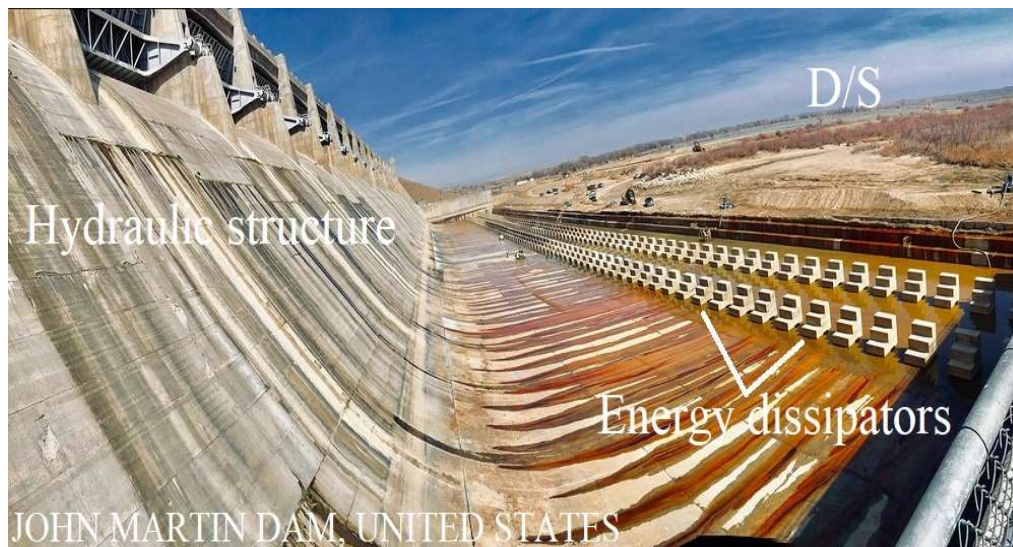


Figure 1.1 Stilling basin downstream of the dam, Jhon Martin, United States

Hager in 1988 [5] emphasised knowledge regarding hydraulic jumps in sloping channels. Underlined their properties and behaviour as a key factor in the development of efficiency dissipation structures for hydraulic engineering applications. This understanding plays a significant role in the design of effective stilling basins, which are commonly used at the bottom of spillways. According to Chow (1959) [6], it is commonly observed that stilling basins are not built in such a way that a free hydraulic jump is fully restrained by the length on a particular apron. This is primarily due to the high cost associated with constructing such a basin. Hence, it is common practice to install accessories in the basin to regulate the jump. The primary objective of implementing such a control mechanism is to effectively decrease the distance covered during a jump, consequently leading to a reduction in the dimensions and expenses associated with the stilling basin. The evolution of hydraulic jump chambers (HJC) or stilling basins can be traced back to the initial designs proposed by Wiggert in 1971[7].

Subsequent modifications were made by Sarikelle and Simon (1980) [8], Wiggert and Erfle (1972) [9]. In their study, Korom et al. (1990) [10] undertook a comprehensive investigation focused on model studies aimed at identifying the most effective design for HJCs (Hydraulic Jump Chambers). The conducted studies encompassed the construction of models of HJCs, wherein different arrangements of baffle blocks were implemented. The primary objective was to evaluate the performance of these models in terms of velocity reduction.

1.2 Hydraulic jump



Figure 1.2 Phenomenon of hydraulic jump occurring downstream of the Hira Kund dam, Odisha, India

In a study conducted by Chow (1959) [6], the hydraulic jump phenomenon was examined. This phenomenon is commonly observed in open-channel flow situations, such as the flow of water in rivers, channels, and spillways. The observed phenomenon exhibits a notable decline in the rate of fluid motion and a sudden elevation in the level of water over a comparatively limited span. In their study, Knight and Shiono (1996) [11] examined the flow dynamics in natural rivers, specifically focussing on the occurrence of hydraulic jumps caused by natural obstacles. The authors delved into the mechanisms and processes involved in the formation of these hydraulic jumps,

shedding light on their significance in understanding the overall flow behaviour in natural river systems. The formation of hydraulic jumps in natural streams is primarily attributed to the presence of natural obstacles. The hydraulic jump phenomenon was achieved by constructing artificial obstacles in the flow path when natural obstacles were not present. The implications of such factors have been found to have a positive impact on the overall health and well-being of aquatic species and their habitats. Figure 1.2 depicts the hydraulic jump phenomenon, which occurs downstream of a hydraulic structure. In his seminal work, Rajaratnam (1967) [12] undertook a comprehensive investigation into classical jumps, delving into the intricate dynamics of velocity profiles and the dissipation of energy. Moreover, the scholarly works of McCorquodale (1986) [13] and Hager (1992) [14] have contributed to the advancement of knowledge regarding hydraulic jump characteristics. These studies have delved into various aspects such as the relationships between sequent depths and the efficiency of energy dissipation. In their recent publication, Padova and Mossa (2021) [15] provided a thorough examination of the historical progression and advancement of studies pertaining to hydraulic jumps, with a specific focus on energy dissipation and the design of stilling basins.

1.2.1 Practical applications of hydraulic jump

The utilization of hydraulics jumps as a means to reduce energy and velocity downstream of a dam or chute, with the aim of mitigating and managing erosion of the channel bed, was examined by Hager (1992) [14]. Hydraulic jumps, a phenomenon extensively studied by Chanson (2004) [16], play a crucial role in flood management strategies. These jumps are employed to effectively control and mitigate the impact of floodwaters. For pollution control purposes, aeration of water is commonly employed in natural channels.

1.2.2 Classical theory of hydraulic jumps

Specific-energy, hydraulic jump, and specific force diagrams are discussed on Figure 1.3. It means that the given specific energy is the lowest at some accurate point. As shown also in Figure 1.3, the water depth y_c in open channel that relates to the minimum specific energy is defined as the critical depth. The water depth y_c in open

channel, which corresponds to the minimum specific energy as depicted in Figure 1.3, is specified as the critical depth. The pre-jump depth y_1 is termed as initial depth of water whereas the post-jump depth y_2 is called the sequent depth or conjugate depth of jump. Figure 1.4 depicts the schematic diagram of the hydraulic jump phenomenon, illustrating the various parameters associated with it.

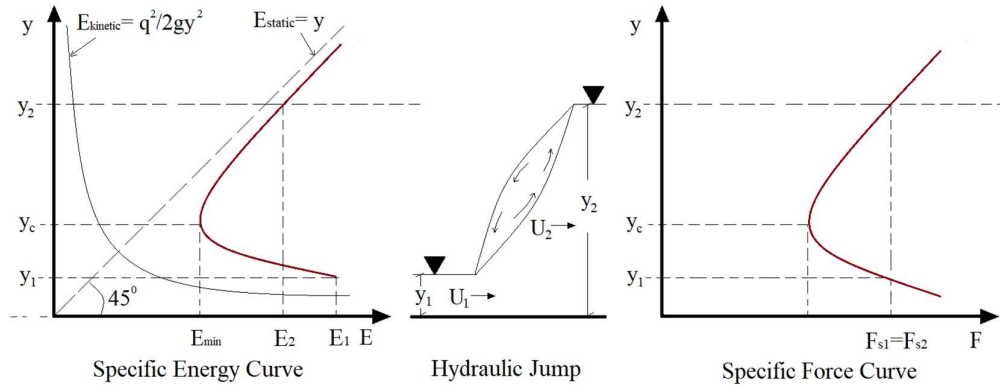


Figure 1.3 Definition sketch of specific energy, hydraulic jump and specific force curve [6]

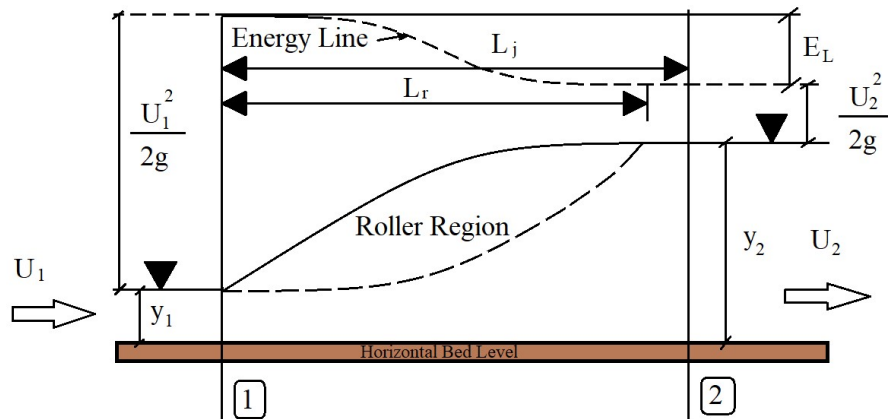


Figure 1.4 Definition sketch of hydraulic jump showing relevant parameters

Rajaratnam (1967) [12] defined the length of a jump (L_j), the range measured as a distance from the toe to the place practically to where the level of water settles, and the mean elevation of water attains its peak. However, it should be noted that this definition can be ambiguous. The length of the roller (L_r) is a very significant factor in studies carried out on hydraulic jumps as it profoundly affects energy dissipation,

flow stabilization and the design of hydraulic systems in general. The length of a roller, as defined by Murzyn and Chanson (2009) [17], is the stretch over which the mean free surface level advances monotonically. Additionally, numerical study conducted by Ghaderi et al. (2021) [18] gives a precise illustration of the exact position where the gap from the take-off point to the initial point of the positive forward movement is known as the roller length of jump.

1.2.3 Types of flows

Henderson (1966) [19] and Chanson (2004) [16] classified the flow on the basis of critical depth (y_c) and the actual depth (y) of water in an open channel. When $(y) > (y_c)$, flow referred to as tranquil or subcritical, for $(y) = (y_c)$ flow referred to as critical and for $(y_c) > (y)$ flow referred to as torrential or supercritical flow.

1.2.4 Hydraulic jump in a horizontal rectangular channel

The analysis of hydraulic jumps typically involves examining the depth of flow prior to and following the jump. Hager (1992) [14] discussed that the Froude number plays a crucial role in determining the flow conditions that result in a hydraulic jump and the equations that govern the classical jump involve the correlation between the tailwater depth and the length of the roller.

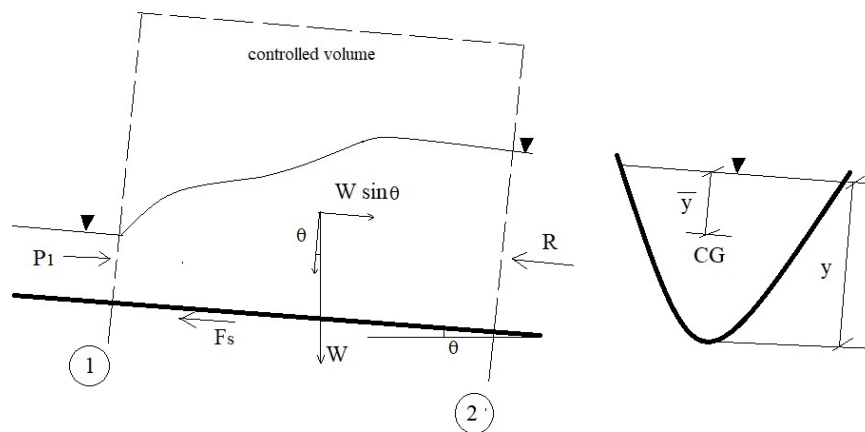


Figure 1.5 Sketch showing controlled volume of fluid

The energy equation is not applicable for analysing hydraulic jump due to significant energy loss and limited understanding of the exact amount of energy loss associated with the phenomenon. The classical theory of hydraulic jump utilises the momentum

principal hydraulic jump le to analyse hydraulic jumps. Figure 1.5 depicts the specific section of the that is being analysed as the control volume.

In studying hydraulic jump, one assumes that there is flow uniformity with hydrostatic pressure in the initial phases of the jump as well as of the final phases of jump. The relative loss as a result of friction on the channel floor is assumed to be small relative to the length of the jump so as to be neglected. Additionally, it is assumed that the channel floor is horizontal in order to eliminate the weight component of the water mass that makes up the jump.

The jump that occurs in most water exercises can be explained with the help of three fundamental equations of mass, momentum and energy conservations.

Continuity:

The relationship between velocities and cross-sectional area and volume flow rate Q can be expressed as follows:

$$A = \frac{Q}{U} \quad (1)$$

Momentum:

With the application of momentum principle to the control volume (in Figure 1.5):

Net pressure force = Rate of change of momentum

$$P_1 - R - F_s + W \sin \theta = M_2 - M_1 \quad (2)$$

P_1 , R is the pressure force at the control surface at section 1 and 2 respectively.

F_s = Shear force acting on the control surface in near the channel bed.

M_1 and M_2 is the longitudinal momentum flux at the beginning and at the ending of the control volume.

The rectangular channel is assumed to be horizontal and frictionless, with a unit width.

Equation (2) can be expressed as:

$$p_1 A_1 - p_2 A_2 = \rho Q (U_2 - U_1) \quad (3)$$

The average pressure can be determined by calculating the pressure at the centroid, denoted as \bar{p} and may be written as $\rho g \bar{y}$, where \bar{y} represents the depth of the centroid below the surface.

On expansion of equation (3),

$$\rho g \bar{y}_1 A_1 - \rho g \bar{y}_2 A_2 = \rho Q \left(\frac{1}{A_2} - \frac{1}{A_1} \right) \quad (4)$$

Assuming unit width of the channel ($b=1$) and substituting $\bar{y} = (1/2) y$, $A = y$, $Q = q$ in above equation

$$\frac{y_1^2 - y_2^2}{2} = \frac{q^2}{g} \left(\frac{1}{y_1} - \frac{1}{y_2} \right) \quad (5)$$

$$\frac{1}{2} * \frac{y_2}{y_1} * \left(1 + \frac{y_2}{y_1} \right) = \frac{q^2}{g y_1^3} \quad (6)$$

On replacing the $\left(\frac{q^2}{g y_1^3} = \frac{U^2}{g y_1} \right)$ with Fr_1^2 (The dimensionless Froude number, denoted as Fr_1 , plays a significant role in open channel flows. This is known as the inertial force to gravity force ratio).

$$\frac{y_2}{y_1} = \left(\frac{1}{2} \right) * \left[-1 + \sqrt{(1 + 8 * Fr_1^2)} \right] \quad (7)$$

According to Tokyay (2005) [20], equation (7) for a horizontal, frictionless and rectangular channel is referred to as the Belanger momentum equation.

Energy:

With the application of energy equation to control volume (in Figure 1.5).

$$E_1 - E_2 = \left(y_1 + \frac{q^2}{2g y_1^2} \right) - \left(y_2 + \frac{q^2}{2g y_2^2} \right) \quad (8)$$

On simplifying the above equation

$$E_L = (y_2 - y_1)^3 / 4 y_1 y_2 \quad (9)$$

1.2.5 Classification of Jumps

Figure 1.6 illustrates the categorisation of jumps based on Froude number. The occurrence of hydraulic jump is influenced by various factors like bed roughness. Subramanya (2009) [21] discussed the five types of jumps.

The Undular Hydraulic Jump, characterised by a Froude Number ranging from 1 to 1.7, exhibits irregularities in its formation and is not well-defined. Turbulences can be observed in the water particles during this phenomenon. A weak jump occurs in a

situation where the velocity of water is relatively small, and such jump occurs prevails when the Froude number is equal to between 1.7 and 2.5.

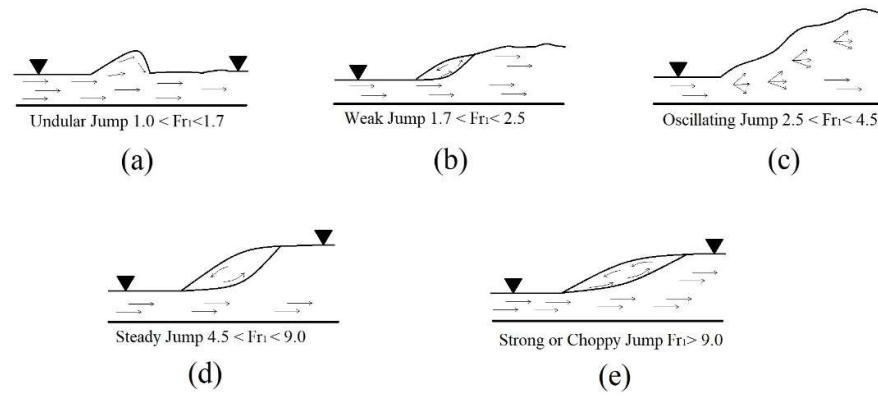


Figure 1.6 Classification of jump based on Froude Number [21]

The oscillating hydraulic jump is formed if an oscillating jet becomes a super critical state where the particles contained in the flow oscillate either in a clockwise or an anticlockwise manner. This type of jump happens when Froude number falls in the range of 2.5 and 4.5. Another familiar type of jump is steady jump in which the particle migrates in one direction with high turbulence and such type of jump is born when the Froude number ranges approximately 4.5 to 9. The case of strong hydraulic jump take place when Froud number is greater 9, from this point the turbulence is higher and water surface elevation is high as compared to steady jump.

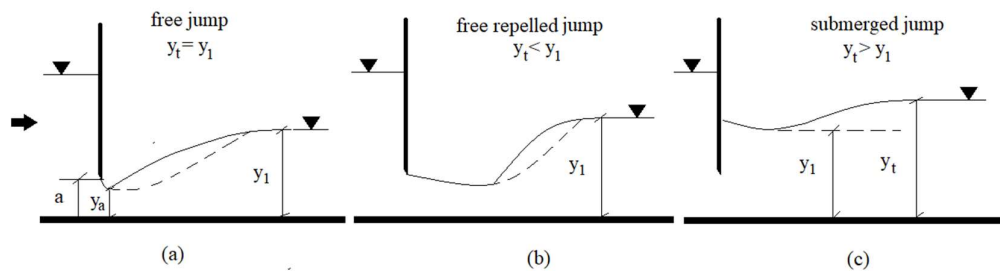


Figure 1.7 Sketch showing different types of hydraulic jump based on depth

Further, Subramanya (2009) [21] discussed types of jumps based on location of jump. Figure 1.7 illustrate the types of jumps based on location of jump. Jump can be defined as free, free repelled and submerged based on tail water depth (y_t) and normal depth (y_1). When tailwater depth equals to the normal depth of water it is referred as free jump. When tailwater depth remains lower than the normal depth it is referred as free repelled jump and for tail water depth greater than the normal depth it is referred as submerged hydraulic jump.

1.3 Computational Fluid Dynamics

Computational Fluid Dynamics (CFD) implements numerical methods and algorithms based on the concepts of fluid mechanics for solving the problems of fluid flow. John von Neumann played a remarkable role in development of algorithms. John D. Anderson (1995) [22] covered the fundamentals of CFD, including numerical methods, grid generation, and applications. Required surface description in terms of its boundary conditions and use when analysing the interaction between liquids and gases in the CFD models. This technique helps design teams reduce the risk of design failures, optimize engineering designs, and gain a competitive market edge. CFD models are based on the Navier-Stokes equations and are solved iteratively either in steady-state or transient modes.

1.3.1 Multiphase models

H.K. Versteeg and W. Malalasekera (2007) [23] provided an introduction of the finite volume method which is the pivotal for the realization of the Volume of Fluid (VOF) in CFD. It also explains the basic of the VOF method and focuses on the steps of develop of the method which are interfacial capturing, estimation of volume fraction and, surface tension forces. The VOF model is a computational multiphase technique for tracking the volume fraction of each fluid in every cell, accurately representing fluid behaviour and interactions (Canonsburg, 2013) [24]. Its primary goal is to capture the interface shape and dynamics between immiscible fluids, efficiently addressing problems like free-surface flows and surface tension effects. The Eulerian model considers each phase as an overlapping continuous medium; the conservation equations of the sub-phases are solved and precise interaction between phases –

momentum, mass, and energy exchanges are accounted for. This method suits systems where both phases significantly occupy the computational domain and interact strongly. The Mixture Model considers the multiphase flow as a single entity, where a single set of momentum equations is solved for the entire mixture and velocities between the phases are taken into account. It simplifies the multiphase flow problem by assuming local equilibrium between phases, making it computationally less intensive than the full Eulerian model.

1.3.2 Turbulence models in computational fluid dynamics (CFD)

The RANS (Reynolds-averaged Navier-Stokes) method is widely used in the field of CFD for the prediction of turbulent flow regimes. RANS that govern fluid motion are time averaged which implies the study focuses on the average flow parameters velocity, pressure etc. Wilcox (2006) [25] discussed different types of turbulence models that is utilized in CFD such as $k-\epsilon$ and $k-\omega$.

Due of its wide range of applications, the $k-\epsilon$ model is widely applied for RANS simulations. It evaluates the value turbulent kinetic energy, k and its rate of dissipation, ϵ , hence the two model equations. This model is based on the assumption that turbulent eddies die out via viscous diffusion and turbulent transport, where the decay is proportional to the rate of dissipation of kinetic energy. The $k-\epsilon$ model is commonly used to solve complex fluid dynamics problems by resolving two distinct equations throughout the simulation. Its performance is comparable to experimental results and other models like the $k-\epsilon$ model. Additionally, it uses a scalable wall-function to enhance solver efficiency and shows increased robustness with fine meshes. This model is commonly used for cost-effective turbulent flow simulations and does not require geometry-specific parameters.

The $k-\omega$ model, a variant of the $k-\epsilon$ model, focuses on the specific dissipation rate of turbulent kinetic energy, making it more suitable for near-wall flows. This tends to include computation of the transport variable k , turbulence kinetic energy and turbulence dissipation rate. The $k-\omega$ model often outperforms the $k-\epsilon$ model in certain scenarios. The software may also offer near-wall treatment solutions and address low Reynolds number problems. This model's advantage lies in its use of the ϵ -equation alongside the ω -equation, facilitating more accurate numerical simulations.

The Reynolds Stress Model (RSM) is a computational approach that calculates the Reynolds stresses directly, which are the key factors causing turbulence anisotropy. Although the computational cost of this model is higher than that of eddy viscosity models, it provides improved accuracy for more complex flows. Spalart Allmaras turbulence model is a turbulence model that was developed by Spalart and Allmaras and is one of the most commonly used models in CFD. This one-equation model is suitable for high Reynolds number flows, as it solves for the eddy viscosity.

Numerous methods are utilized in the discipline, such as Large Eddy Simulation (LES), Direct Numerical Simulation (DNS), and Detached Eddy Simulation (DES). LES is commonly used to simulate turbulent flows by directly resolving large-scale eddies. On the other hand, a Sub Grid Scale (SGS) model is used to close the equations for smaller scale eddies. Unlike approaches based on rheological models for flows, including all of the turbulence in the geometrical grid, LES is convenient for the design of complex turbulent flows which exhibit a large range of scales, from the large systems down to the smallest vortices. DNS, or Direct Numerical Simulation, is a computational method that simulates turbulent flows by numerically solving the Navier-Stokes equations, eliminating the need for turbulence modelling. This technique allows for precise simulation across various scales, from large structures to minute eddies, without requiring sub grid-scale models. DNS is considered the most accurate technique for turbulent flow simulation, capturing all intricate flow field details, including small-scale features not accurately represented by traditional turbulence models. The DES model integrates aspects of RANS and LES models, facilitating a seamless transition from RANS to LES in regions with well-resolved turbulence. This approach is particularly effective for flows featuring separation and recirculation.

The detailed discussion on transport equations involved in above discussed models are done in Chapter 3 of the present thesis.

1.4 Need of the study

At present, the comprehension towards the utilization of CFD in the area of hydraulic engineering is somewhat vague. The need of the study is to offer valuable insights to

field engineers on the design of hydraulic energy dissipators using computational fluid dynamics (CFD) modelling. The aim is to provide these insights in a cost-effective manner. Additionally, it has been determined that the hydraulic jump is the primary phenomenon for energy dissipation in the discipline of hydraulic engineering. However, it is worth noting that this topic has not been extensively studied using Computation Fluid Dynamics. The following are a list of novel and rarely studied gaps in the field of Computational Fluid Dynamics and the Hydraulic Jump phenomenon.

Although a lot of research has been done on hydraulic jumps, there are still major gaps specifically in learning how strips of roughness on the bed affect hydraulic jumps. Research on roughness so far usually covers general impacts but does not look at how the shape, size and alignment of roughness elements in stilling basins influence water flow. The behaviour of various strip macroroughness shapes, in particular triangular, trapezoidal, semicircular and novel forms, has not been thoroughly analysed together with their aspect ratios. Further, no studies have thoroughly investigated the role of the layout and positioning of roughness within stilling basin type hydraulic jumps. These problems are compounded, since past numerical studies often fail to consider how different boundaries affect the action of hydraulic jumps over rough areas and how this affects the flow, energy loss and computer calculations. In addition, not much is known about the influence of boundary conditions on successful convergence, the quality of results and how long the simulation takes. The absence of clear evaluation methods decreases how dependable and effective CFD models are for hydraulic jump research, mostly in cases of complex configurations or rapid turbulence. Dealing with these overlooked items is crucial for making advanced and affordable numerical frameworks for stilling basin design. It is because of these vital parts that we need to use a combined approach of studying different geometries and using advanced computer modelling to improve energy management and flow control in hydraulic structures. Motivated by these gaps, the present study employs high-resolution Computational Fluid Dynamics (CFD) simulations using Ansys Fluent with the Volume of Fluid (VOF) method and multiple turbulence models to evaluate hydraulic jump characteristics over smooth and artificially roughened beds. Through mesh sensitivity analysis and model validation against experimental data, the study aims to provide detailed insights into optimizing

roughness configurations for enhanced energy dissipation and flow stability, thereby supporting the design of more efficient and cost-effective stilling basins

1.5 Objective of the study

The objective of this study is to investigate the characteristics of hydraulic jumps over smooth and macrorough beds in a prismatic rectangular channel located downstream of a hydraulic structure operated by a sluice gate. The following objectives are set to numerically study the hydraulic jump characteristics under varying channel conditions. The present study aims to investigate multiple aspects of hydraulic jump behaviour under different conditions. Specifically, it focuses on studying the effect of various flume configurations on the characteristics of classical hydraulic jump. Additionally, it examines the effect of various artificial strip macroroughness on the characteristics of free and submerged hydraulic jump. Further, the study explores the influence of the height to base width ratio of macroroughness and their possible arrangement on free and submerged hydraulic jump characteristics. Lastly, it investigates the effect of the height to wavelength ratio of macroroughness on free and submerged hydraulic jump characteristics.

1.6 Outline of thesis

The present document is structured into five chapters, each addressing a specific aspect of the study. CHAPTER 1 gives a brief introduction to the hydraulic jump phenomenon, numerical modelling techniques, the necessity of the study, its objectives, and the overall scope of the present work. CHAPTER 2 provides knowledge regarding numerical and experimental investigations of hydraulic jumps over both smooth and macrorough beds, drawing upon existing literature. CHAPTER 3 elaborates on the approach utilized for the current study, detailing the methodology and the process through which the numerical models were validated. CHAPTER 4 presents the results and discussions arising from the numerical simulation of the hydraulic jump phenomenon under various conditions. Finally, CHAPTER 5 summarizes the conclusions drawn from the present study and outlines the scope for future research work in this domain.

CHAPTER 2: LITERATURE REVIEW

2.1 General

In past wide range of literatures are available on hydraulic jump analysis which are based on theoretical, physical and numerical investigations. This section tried to explore the current state of knowledge.

Rajaratnam (1967) [12] offered exhaustive insight of many different kinds of hydraulic jumps such as the classical hydraulic jump, submerged jump, forced jump, jump in sloping channel, pre-entrained jump and jump in non-rectangular channel, jump in conduit, jump in non-prismatic channel, and jump in stratified flow. He used Prandtl type pitot tube to measure the forward velocity and from the observations, the velocity profile of a hydraulic jump was found to be of plane turbulent wall jet. He emphasized the importance of taking into account the longitudinal velocity gradient and the bed shear stress distribution in the jump region.

France (1981) [26] examined hydraulic jumps occurring in a rectangular gradually varied channel. In his study he took into consideration the divergence angles of up to 8.7° and no bottom slope. The objective of such jumps was to compare it with the conventional hydraulic jumps that occur in prismatic channels. He derived theoretical equation to compute sequent depth ratio factoring the influence of effective Froude number and side pressure forces. He reported that the parameter was invariant with the angle of divergence but when angle of divergence was increased a lot more fluctuation in the jump stability was experienced. He concludes that energy dissipation in diverging hydraulic jumps is greater than energy dissipation in converging hydraulic jumps and this difference increases with the initial Froude number. Furthermore, the height of the jump was also shorter by up to 40 % when compared to what is referred to as the classical hydraulic jump. Unlike a parallel channel, it is observed that the relative energy loss can increase up to a maximum of about 12%. The overall length of the hydraulic jump that was observed was relatively shorter than the length calculated using the theoretical model. He concluded that, by using diverging rectangular channels, energy can be well discharged through hydraulic jumps.

Hughes and Flack (1984) [27] conducted various experiments in a horizontal rectangular flume with smooth sidewalls to investigate the characteristics of hydraulic jump over rough and impervious beds. They used 5 long test beds, one plain and 2 strip roughened beds and 2 densely packed gravel beds. The maximum relative roughness value was set to 0.9. The researchers recorded flow rate, upstream depth, tailwater depth, and jump length from roughly 200 hydraulic jump observations. They noted that the behaviour of the shear stress and sequent depth ratio over a roughened bed varied with the initial Froude number in different ways. In particular, the shear stress grew with the decrease of the sequent depth ratio when the first Froude number augmented.

Hager and Bretz (1986) [28] analysed the important hydraulic flow characteristics described in hydraulic jumps that take place at both positive and negative steps in a rectangular, horizontal, and prismatic channel. They paid more attention to these factors with a view of establishing the correlation between length of jump and these phenomena. They stated that the shape of the step was not crucial as it depends on the transition from supercritical to subcritical flow at the step. Consequently, the sequent depth ratio was highly dependent on the height of the steps as well as the approaching Froude number. Moreover, the evaluation indicated that the degree of the tailwater depth was nearly six times more variable in the case of the negative step than the flow depth. Hydraulic jumps taking place at negative steps are characterized by a higher dynamic in the adjustment of the depth of the tailwater and the Froude numbers in which efficiency might be maximized. Therefore, the tested hydraulic jump over positive steps proved to be more stable as compared with that over negative step.

Hager and Wanoschek (1987) [29] investigated hydraulic jump in a channel of semi triangular cross-section and compared it with a rectangular cross section. They described that the horizontal velocity profiles and the free surface forms are characteristically different in the case of the triangular channel. However, here the sequent depth ratio was observed to be lesser than that identified in the case of the rectangular channel. It was found that the efficiency of energy dissipation in the conical channel was approximately 30% higher in comparison with one realized in a flat channel. Further it was observed that the distance jumped over by the mass was of

equal to one half of the length of the rectangular channel. Despite the fact that the triangular channel was 1.39 times shorter than the rectangular one, the volume of the hydraulic jump was 30% more. A wider surface width at the onset of the jump was observed in the triangular channel, and may cause limitation in the available lateral space. The hydraulic jump in the triangular channel was less sensitive to small changes in the discharge but more sensitive to small changes in the tailwater compared to the rectangular channel.

Hager (1988) [5] examined the hydraulic jump in a rectangular channel of invariably varying width. He contemplated a steep slope at the channel bottom in the upstream portion of the canal and a lower slope toward the downstream portion. He divided various jump types (A, B, C, D) on the basis of hydraulic jump with sloping as well as level part of the channel. He worked out several equations for finding sequent depth ratio, roller length and the initial Froude number and relative static head regain. The equations were ideal for the need of designing. Further, it expanded on the hydraulic jump and how it is useful in controlling the energy dissipation of the flow as one of the critical avenues for controlling flow and eradicating erosion in stilling basins. It was also identified that the work done in the case of the formation of a hydraulic jump in the classical way had the highest efficiency.

Long, Rajaratnam, et al. (1991) [30] investigated the hydraulic jump in a horizontal rectangular flume of width 0.47m, height 0.52m and length 7.5m. They used Froude numbers between 4 and 9 in their study. They developed the replica of flow regime near the wall by employing high speed photographic technique, where they took about 2000 frames/s. Jump was subsequently disaggregated into several parts based on inlet Froude numbers as well as timing partitions. In the multiple graphs they drew the changes taking place in the surroundings of the vortices. These multiple graphs denote the locations and sizes of these vortexes based on the extruded analogy of the vortex dominated zone and correlate to the change in Froude numbers. They said that it is crucial to shed light on the vortex pattern of hydraulic jump especially in these tremendously huge structures. Besides providing a complete picture of vortex action inside them, a precise description of such action is useful when it comes to making

more accurate predictions of jump behaviour and developing the most effective hydraulic structures.

Rajaratnam (1995) [31] investigated several methods of quantifying hydraulic jumps. Such techniques included direct methods, which engaged tools like velocity probes and pressure transducers, and indirect methods that embraced methods like flow visualization and the surface profiles. It features not only measurement approaches and problems and limitations related to it but also shows how flow conditions, channel geometry, or turbulence might influence hydraulic jump measurements.

Alhamid and Negm (1998) [32] proposed an easy mathematical model to predict the sequent depth ratio of hydraulic jumps in abruptly widening rectangular channels. From the collected experimental data and analytical tool, they analysed the cause-effect relationship of different parameters associated with hydraulic jump. They developed an equation based on the application of one-dimensional momentum and continuity equations adjusting the classical Belanger equation by the effects of sudden expansion through Froude number, and finally they mentioned that derived equation provides the simplest and more effective method to predict sequent depth ratio compared to other models. Despite the fact that the algorithm of its application is very simple, it can be helpful in designing such hydraulic structures as stilling basins. Their results indicate that the new models would be ‘design friendly’ indicating that design of improved models is highly achievable. The most significant impact was recorded concerning the sudden expansion they realized had a negative impact on stability.

Wu and Rajaratnam (1996) [33] explored the behaviour of the hydraulic jump from the wall jet flow structure to the semilogarithmic velocity profiles of open channel flow. They tested in a parallel flow channel with a flat, slippery bottom and transparent sidewalls with Froude numbers from 3.87 to 10.48. They observed the velocity profile at first moment seemed to behave like wall jet with maximum at the bed level. They concluded that during transition from hydraulic jump to open channel flow, the bed shear stress was decreased along with other velocities. The minimum bed shear stress declines with an increase in Froude number and becomes near zero at a certain Froude number which suggests the possibility of flow separation from the bed. About the

centreline there is dissipation of energy which dissipates the flow intensity until it gets into the intensity of the open channel flow. They further pointed out that the length of the transition zone was roughly ten times the tailwater depth. The extent of this transition region was found to be adequately characterized by the Froude number and the depth of the tailwater.

Ead and Rajaratnam (2002) [34] studied the dynamics of hydraulic jumps on particular types of channels called corrugated beds, during a series of experiments in the laboratory. They proceeded with experiments at four Froude numbers of 4 to 10 with relative roughness range of 0.25 to 0.50. They pointed out that depth of tail water needed for the formation of jump on corrugated bed was relatively small when compared to the depth needed for the formation of jump in smooth beds. They said that that hydraulic jumps on corrugated beds were almost 50% the length of that in smooth beds and also observed that the shear stress was about 10 times as much in corrugated beds as in smooth beds. Axial velocity profiles in the jump were also compared to the jump in the smooth beds but with some disparity. The boundary layer thickness relative to the length scale of the velocity profile grew faster on corrugated beds compared to smooth beds. They indicated that hydraulic jumps on corrugated beds could be highly effective in dissipating energy, making them an attractive option for energy dissipators under hydraulic structures. However, they suggested further exploration of the effects of corrugation amplitude and wavelength.

Tokyay (2005) [20] investigates the effect of corrugated channel beds on hydraulic jumps characteristics such as jump length, energy dissipation, tailwater depth, and jump location through experimental research. He used two corrugated aluminium sheets with different wave lengths and heights and a rectangular channel of certain dimensions. He disclosed that the depth of the tailwater needed to develop the hydraulic jump on corrugated beds was roughly 20% less than on smooth beds. The recorded height of the jump was approximately 35% lower over corrugated beds than the height over smooth beds. It was found that jumps, made on corrugated beds, exhibit higher energy dissipation, thus making them suitable for energy dissipation. He confirmed that corrugations can be an effective alternative to traditional structures like baffle blocks and sills in stilling basins.

Omid, Esmaceli Varaki, et al. (2007) [35] proposed a theoretical and experimental investigation of the hydraulic jumps in trapezoidal channels with gradually expanded stilling basins. They were able to advance the basin by enhancing the width of its bed in the stream wise direction. Angles of expansion ranged from 3 to 9 degrees and trials were conducted on three different side slopes. They conducted experiments in a laboratory setting with a channel equipped with adjustable side walls to simulate various angles and side slopes. They measured upstream and downstream depths, jump length, and flow conditions, with Froude numbers ranging from 3 to 10. They have presented a one-dimensional momentum equation, which could be used to predict the sequent depth and energy loss in a trapezoidal channel. They reported the surface profile of the jump was approximated by a quarter-elliptical shape. The stability of the hydraulic jump improves with increased expansion angles, especially at higher Froude numbers. Increasing the expansion angle reduces the sequent depth and jump length while increasing energy loss in the jump. Similarly, decreasing the slope of the basin walls leads to smaller sequent depth but larger jump length and energy loss. Divergence of the basin results in higher energy dissipation compared to hydraulic jumps in rectangular or trapezoidal sections with unchanging dimensions.

Carollo, Ferro, et al. (2007) [36] investigated hydraulic jump in a rectangular channel. They assessed rough beds in form of artificial and natural stones. These stones were characterised by different grain size distributions and the grain size values varied between 0.46 to 3.20 cm. They carried out only experimental work to validate the new solution of the momentum equation for the sequent depth ratio and to assess the empirical relation for the roller length. They also discovered that at some Froude number and particle roughness height the sequent depth ratio was less than the sequent depth of a smooth channel bed.

Chanson and Gualtieri (2008) [37] conducted experiments in two geometrically similar channels with a scaling ratio of 2:1 to study how air entrainment scales. They observed void fraction distributions and bubble count rates, which followed a diffusion model in the larger channel but showed scale effects in the smaller one. Lower Reynolds numbers in the smaller channel resulted in greater air detrainment, leading to less overall aeration. They revealed that dynamic similarity in hydraulic jumps was

affected by multiple parameters, such as Froude, Reynolds, and Morton numbers, making it difficult to perfectly model air entrainment at different scales. The small-scale models may underestimate air entrainment characteristics, providing insight into the limitations of physical modelling in hydraulic engineering. The smaller hydraulic jumps possessing lower air entrainment and higher air detrainment phenomenon showed overall lower aeration of the jump roller in comparison with the larger hydraulic jumps. The measurements of the dimensionless bubble count rates were significantly lower in the smaller channel particularly in the mixing layer. The also identified an advective-diffusion shear layer in hydraulic jumps, but for Reynolds numbers below a critical value, air entrainment was weak, and detrainment processes dominated. The inability to achieve dynamic similarity due to differences in Reynolds and Morton numbers highlights challenges in scaling hydraulic jumps and maintaining accurate modelling of air entrainment processes across different model sizes.

Murzyn and Chanson (2009) [17] primarily focused on experimental studies of hydraulic jumps with Froude numbers ranging from 3.1 to 8.5. They examine the chaotic motions between large-scale eddies and the free surface and the application of acoustic displacement sensors and phase-detection probes for free-surface oscillations and two-phase flow characteristics. They observed free-surface fluctuations increases with the Froude number, especially in the roller region of the jump. The turbulent fluctuation levels peaked in the first half of the roller, with the peak increasing with larger Froude numbers. For instance, the maximum fluctuation was reported about 1.5 times the inflow depth for a Froude number of 8.5. They found acoustic displacement meters to be an effective tool for measuring free-surface dynamics in hydraulic jumps, with their time-averaged readings closely corresponding to the division of the flow domain between the isolated turbulent shear layer and the upper free-surface layer.

Abbaspour, Hosseinzadeh Dalir, et al. (2009) [38] examined the effect that wave steepness of the corrugated wall had on hydraulic jump features like the location of the free surface, the fluid velocity and the shear stress as well as energy dissipation rates. They varied initial Froude numbers from 3.8 to 8.6 for six different quantities of corrugated beds, where each of those values corresponded with a set wave steepness ranging between 0.286 to 0.625. They noted that the flow and the length of jump are

substantially lower over the corrugated bed than in the smooth channel bed. The depth of the tailwater depend on the Froude number although the wave steepness does not have any effects. On the corrugated bed, the normalised boundary layer thickness was found to be 0.57. The stress distribution in the corrugated beds was seen to be about 10 times of smooth beds. They reported that the application of corrugated beds at stilling basins can be useful in the dissipation of hydraulic jump energy.

Abbaspour, Farsadizadeh, et al. (2009) [39] evaluated hydraulic jumps on corrugated beds using computational fluid dynamics (CFD) and turbulence modelling. They used the two-dimensional RANS equations for flow predictions, VOF to capture free surface. They utilized two equation k- ϵ turbulence models to simulate the turbulent characteristics of the hydraulic jump. They found good agreement between simulated and experimental data with relative errors between 1% and 8.6% for water surface profiles. The jump length predictions had errors ranging from 1% to 12%. The RNG k- ϵ model provided a slightly better match with experimental data in contrast to the standard k- ϵ model. They found corrugations on the bed effective to reduce the length of the hydraulic jump and tailwater, mitigating cavitation risks by keeping crests below the flow.

Debabeche, Cherhabil, et al. (2009) [40] used both theoretical and experimental methods to investigate the impact of the channel slope on sequent depth ratio in an open channel triangular channel with six positive slopes of the channel of 0%, 2%, 5%, 10%, 15% and 20%. they developed an expression for inflow Froude number in terms of sequent depth ratio and channel bed slope. From this they obtained an expression which related the sequent depth ratio, as well as the length of the jump, to the channel slope. A correction coefficient was introduced to account for discrepancies between the theoretical and experimental jump volumes. This coefficient was reported invariant and hence not affected by the Froude number and the channel slope. They confirmed the published data that even the slope of the triangular channel remains affecting the sequent depth ratio of the hydraulic jump.

Elsebaie and Shabayek (2010) [41] investigated hydraulic jump on various shapes of strip corrugation to understand jump characteristics. These shapes comprised of

sinusoidal, triangular, and trapezoidal with two side slopes and rectangular. They used variables like sequent depth, length of jump, depth deficit factor, velocity at the different sections of the channel and the bed shear stress. They considered Froude number ranging from 3 to 7, and the relative roughness values of 0.36 and 0.72 in their study. They showed that the amount of jump length for corrugated beds was 50% shorter than of that for the smooth beds. This was due to the fact that, because of the bed's corrugations, the bed shear stress increased. Comparing the integrated bed shear stress on corrugated beds and on smooth beds, the latter was reported to be more than 15 times less than the former. The dimensionless depth deficit, which reflects the difference between classical and corrugated bed jumps, was consistently around 0.37 for all bed shapes. It was ascertained that the shape of the corrugation had a very small effect on the hydraulic jump behaviour in spite of constant amplitude and wavelength of the corrugation. They confirmed that the corrugated beds have ability to dissipate energy below hydraulic structures.

Chachereau and Chanson (2011) [42] investigated the morphology of free-surface oscillations and air-water gas-liquid interfacial characteristics in a hydraulic jump roller. They carried out investigations in a horizontal rectangular channel for comparatively low Froude number. They normally employed non-intrusive Acoustic Displacement Meters (ADM) for the real-time measurement of free-surface fluctuations. Also, they used several numbers of Acoustic Doppler Velocimeters (ADV's) to show the turbulence mechanisms of the hydraulic jumps with great accuracy. They determined that the mean free surface profile had a clear configuration in agreement with the visual and photorealistic parameters. The violent character of the fluctuations which formed hydraulic jumps with certain fluids was reported to be anisotropic, especially in the neighbourhood of the free-surface. They identified that the amplitude of turbulence fluctuations strongly dependent on the Froude number of the incoming flow.

Valiani and Caleffi (2011) [43] analysed the conservation of both linear and angular momentum within diverging channel, particularly in turbulent flow conditions, and developed an analytical solution to describe the flow profile and sequent depths in a hydraulic jump. They investigated the interaction stresses between the roller and the

mainstream, revealing that these stresses increase sharply at the start of the jump before gradually decreasing, consistent with turbulent jump behaviour. They developed equations for calculating the flow's free surface profile, mainstream and roller thicknesses, and power losses.

Tokyay, Evcimen, et al. (2011) [44] carried out experiments to study the impact of corrugations and prismatic roughness elements on hydraulic jumps, focusing on characteristics like jump length, tailwater depth, and energy dissipation capacity. They covered the entire basin length and with prismatic roughness elements arranged in either strip or staggered configurations. They reported that roughness elements significantly reduced the hydraulic jump length. More specifically, corrugations reduced it by about 35%, strip roughness by 40%, and staggered roughness by 35-55%. Corrugations lowered the tailwater depth by 20%, strip roughness by 5-13%, and staggered roughness by 7-15% compared to a classical hydraulic jump. Roughness elements also improved energy dissipation, with an increase of 3-10% over classical jumps which makes them viable alternatives to traditional stabilization devices like baffle blocks and sills.

Afzal, Bushra, et al (2011) [45] presented a detailed analysis of the flow structure of a turbulent hydraulic jump in the wide channel with a rough bed. They focused on dividing the flow into inner and outer layers and using depth-averaged momentum equations to analyse the jump in case of rough beds. They reported that the bed roughness only passively influences the flow in the outer layer. The jump characteristics such as conjugate depth ratio, length of jump, and length of roller can be universally described by relations that were independent of the bed roughness when expressed in terms of the effective upstream Froude number. They established several new empirical relationships for roughness drag based on relative roughness.

Chanson (2012) [46] used field data, including measurements from natural and artificial channels, to test theoretical solutions for dynamics of hydraulic jumps, particularly for irregular channels. He highlighted that the shape of the channel and bed friction significantly impact flow properties. In non-rectangular channels, the effects of flow resistance yield a smaller ratio of conjugate cross-sectional areas for a

given Froude number. They reported that field data showed deviations in the Froude number calculation by up to 74% from traditional methods, depending on channel shape and flow resistance.

Tokyay and Velioğlu (2012) [47] examined the impact of various components of bed roughness on the properties of jumps. They stabilized jump locations and reduced stilling basin lengths using alternative roughness elements, such as corrugations, strip, staggered roughness elements, and gravels. According to them, corrugations cause a reduction in jump length by between 20% and 25%, gravels by 20%, strip roughness by 30%, and staggered roughness by between 35% and 55% compared to the classical hydraulic jump. Also, Roughness elements increase energy dissipation by 3-15% more than classical jumps, with gravels being the most efficient at inducing energy loss. The roughness types effectively stabilized jump locations, making them viable alternatives to traditional baffle blocks and sills. They suggested that incorporating roughness elements like gravels and corrugations can significantly enhance the performance of hydraulic jumps in terms of reducing jump length and increasing energy dissipation.

Samadi-Boroujeni, Ghazali, et al. (2013) [48] concentrated in order to determine the influence of triangular corrugated beds on the parameters of hydraulic jumps in a rectangular channel. They conducted 42 tests in a 12-meter-long, rectangular flume using six different triangular corrugated bed configurations. The tests were performed for Froude numbers ranging from 6.1 to 13.1. The authors pointed out that corrugated bed minimizes the conjugate depth by 25% and the hydraulic jump length by a 54.7% compared to smooth beds. They observed that the shear stress on the corrugated bed was about 8.5 times more than that of the smooth beds. The relative height and shape of the corrugations have no significant effect on reducing both tailwater depth and jump length, indicating that no specific corrugated configuration could optimize both parameters simultaneously. They suggested that optimizing both depth reduction and jump length with a single corrugated sheet is not feasible.

Abbaspour, Farsadizadeh, et al. (2013) [49] discussed possible approaches for using soft computational modelling like artificial neural networks and genetic models to predict some features of hydraulic jumps, for instance, the position of the free surface

and energy dissipation on corrugated beds. They conducted 123 experiments on hydraulic jumps formed on a corrugated polyethylene bed to gather data. The experimental setup featured various combinations of Froude numbers and bed roughness heights, among other variables. Using these data, they developed models with artificial neural networks and genetic modelling, intending to estimate the jump depth, length and energy dissipation employing Froude number and dimensions of corrugation. Their analysis revealed that the artificial neural networks model provided better accuracy compared to the genetic model, especially in estimating jump characteristics. The artificial neural networks model was able to achieve higher correlation coefficients and lower errors. The genetic model, while practical for producing mathematical relationships, was found to be less precise in comparison.

Xiang, Cheung, et al. (2014) [50] proposed a comprehensive numerical modelling approach to simulate the complex flow dynamics in hydraulic jumps, focusing on air entrainment and bubbly flow. They combined a Volume-of-Fluid (VOF) approach with the Multiple-Size-Group (MUSIG) model, using a Eulerian–Eulerian multi-fluid framework to enable simulation of air and water interaction, including the entrainment of air bubbles at the jump toe and their dispersion in the flow. They reported that model predicted the location and rate of air entrainment accurately, aligning well with experimental data from previous studies. Also model effectively captured the flow patterns in jet region, shear layer, and recirculation region. regions, particularly the velocity profiles and turbulent roller formation. They reported that bubbles accumulate in the recirculation region, and their size reduces downstream due to breakage by turbulence. Despite its accuracy, the model showed limitations, especially in predicting downstream bubble frequency and velocity near the free surface.

Ahmed, El Gendy, et al. (2014) [51] focused on how corrugated beds affect the properties of hydraulic jumps specifically for submerged jump. They conducted experiments in a laboratory using a flume with a streamlined-lip gate to develop the required supercritical flow. They used aluminium triangular sheets to create the corrugated bed. The corrugation dimensions were 40 mm in height and width with a side angle of 45° . Thirty overall experimental runs were conducted in total and

covering both smooth and corrugated beds with recorded minimum Froude number of 1.68 and a recorded maximum of 9.29. They reported that the relative sequent depth over corrugated beds reduces compared to smooth beds. An optimal spacing of 12 cm between the corrugations resulted in a reduction in sequent depth by approximately 15–17%, depending on the Froude number. The submergence depth reduces by 18–38% over corrugated beds compared to smooth beds. The depth of the water downstream also decreased by up to 15% at the optimal corrugation spacing compared to the smooth bed. It was noted that the bed length was reduced by a maximum of 28% on the corrugated beds. The efficiency of the hydraulic jump increased by up to 50% for corrugated beds compared to smooth beds, with an optimal spacing of 12 cm providing the maximum efficiency.

Esmaili Varaki, Kasi, et al. (2014) [52] conducted a series of experiments in a flume with various bed slopes and diverging angles to study the parameters of the hydraulic jump including sequent depth, jump length, energy loss at the jump under various Froude numbers. They developed a momentum-based theoretical model to predict the sequent depth ratio and compare it with experimental data. The model assumes parabolic water surface profiles and neglects frictional forces. They showed that increasing the adverse slope and divergence angle reduces the sequent depth and jump length while increasing energy loss compared to classical hydraulic jumps. The sequent depth decreases by up to 51%, and the jump length by up to 38% and energy loss increases by up to 23% with the highest divergence angle and bed slope.

Chanson and Shi (2015) [53] presented a comprehensive analysis of the current obstacles and notable progress in hydraulic engineering, with a specific focus on the environmental aspect using experimental and analytical methods. They introduced more varied aspects and including elaborate features of water pertaining to water systems, temporal and spatial scales, water-solid, water-air, and water-biological life. They emphasized the need to create technical solutions for overcoming of the tackling hydraulic problems in the environment.

(Kateb, Debabeche, et al. (2015) [54] investigates the hydraulic jump in a sloped trapezoidal channel through theoretical and experimental methods. They conducted

experiments in a trapezoidal channel at five different positive slopes ranging from 0% to 2% for wide range of Froude numbers from 3.5 to 14.5. The experimental data helped refine the theoretical relation. They derived the theoretical relationships or expressions in terms of the inflow Froude number and do sequent depth ratio, channel slope or steepness. They concluded that the sequent depth ratio increases with an increase in the channel slope.

Bayon-Barrachina and Lopez-Jimenez (2015) [55] put forward a computational fluid dynamics (CFD) model that utilises the Open FOAM platform to design hydraulic jumps taking place in horizontal, smooth and rectangular prismatic channels in 3D manner. They used three turbulence models including the standard $k-\epsilon$, RNG $k-\epsilon$ and the SST $k-\omega$ to encounter the turbulences. They used the Volume of Fluid (VOF) model to represent the interface between two immiscible fluids. They realised that through integrating the CFD modelling method in simulating hydraulic jumps, that the basic hydraulic jump parameter inclusive of water surface profile, sequent depth ratio and length of jump have been determined with a very good accuracy of more than 98 percent.

Kim, Choi, et al. (2015) [56] studied the hydraulic parameters of hydraulic jump using a fixed weir and a sluice gate-type movable weir in a rectangular channel. They used energy dissipators in the downstream of the hydraulic structure in order to sink high rates of kinetic energy and protect the channel bed. They reported that by placing energy dissipators at the sluice gate, energy dissipation can be reduced by more than 52% compared to those without energy dissipators. It was also ascertained that the hydraulic jump lengths of sluice gate type movable weir and fixed weir are of equal length. Movable weirs of sluice gate type can establish flow regions of high velocity and can experience bed scouring. They developed an improved design of movable weirs based on improvements in the design criteria for riverbed protection.

Luo (2015) [57] examined the hydrodynamic characteristics of a submerged hydraulic jump. He employed numerical, experimental, and analytical approach to probe three features including the fluid depth, the turbulent primary velocity, the shear stress and turbulent kinetic energy. He provided such alternatives in mathematical representation

for factors such as water depth, primary turbulent velocity, the shear stress as well as the kinetic energy of turbulence for both fixed and movable beds. He identified three eddy zones, influenced by forward and backward velocities, scoured holes, and adverse pressure gradients which significantly affect the flow structure in submerged hydraulic jumps. Also, he suggested that numerical models using standard $k-\epsilon$ turbulence models need adjustments for certain parameters, particularly in higher Froude number inflows.

Wang and Chanson (2016) [58] understood the scale effects of hydraulic jumps, particularly in down-scaled physical models, as well as the behaviour of free-surface dynamics, air entrainment, and turbulence. They conducted experiments on hydraulic jumps with various Froude and Reynolds numbers, investigating their effects on parameters such as surface deformation, bubble-turbulence interactions, and air entrainment rates. They concluded that while self-similar behaviours exist for certain flow properties, many key turbulent and air entrainment processes are scale-dependent. Full-scale conditions are necessary to accurately capture these effects, particularly for bubble dynamics and turbulence intensity.

Mahtab and Sattari (2016) [59] confirmed the M5 Model Tree over rollers length and sequent depth of the smooth and rough beds. The statistical results of the modelled tree on the M5 Model Tree, ANN and the empirical-based equations for sequent depth ratio were then compared. Regarding the plans of inference, they identified that the ratio of complexity level of the sequent depth proposed by the M5 Model Tree is straightforward and accurate.

Hafnaoui, Carvalho, et al. (2016) [60] performed an experimental investigation to model the jump in a sloping rectangular channel. They managed to produce a MATLAB model operating under the Saint-Venant equations and the TVD MacCormack explicit scheme which was used to predict the location of hydraulic jumps both in rectangular and triangular channels. Beyond that, they obtained correlations to express the errors in hydraulic jump positions; they illustrated the dependence of predictions on both slope and Froude number changes. Both of them stressed that the availability of the model for the prediction of sediment discharge is

valuable, however, adjustments for the slope and downstream conditions are critical for accurate calculations especially for sloped triangular channels.

Sauida (2016) [61] employed Artificial Neural Networks and multiple linear regression model for predicting length of submerged hydraulic jumps downstream of multi-vent regulators. They discovered that Artificial Neural Networks had a slightly higher coefficient of determination and so predicted more accurately than the multiple linear regression model models with 12% improvement. The ANN model results in improved modelling of the length of the hydraulic jump as the influence of the gate operation changes from case to case.

Abbaspour, Parvini, et al. (2016) [62] gave a realistic model to solve the problem of erosion of beds by the provision of buried plates in the channel bed. They placed buried plates on the range of distances from the apron in the open channel with both slope of the horizontal and reverse bed. They also discovered that through the proper positioning and orientation of the plates, scouring was well controlled and reduced. Besides, while comparing the results obtained in case of a single buried plate with that of two buried plates, they found that the effectiveness of the two buried plates placed at 30 cm and 45 cm below the non-erodible bed was significantly higher than the single buried plate in reducing the depth of scouring.

Hassanpour, Dalir, et al. (2017) [63] performed experiments for hydraulic jumps at the zone of transition from a wide rectangular channel to wider, lozenge-shaped channel with divergence ratios of 0.4, 0.6, 0.8, and 1. They examined the post jump depth, the length of jump, length of roller and by changing the inlet Froude numbers in the range of 6 to 12. They stated that the relative length of hydraulic jump is found to depend on the magnitude of relative Froude number and divergence ratio. However, when the ratio of roughness elements increase, the length of the jump reduced as observed earlier. On the other hand, the further apart is the divergence ratio, the greater is the relative energy loss.

Bejestan and Hajibehzad (2018) [64] conducted an experimental study of hydraulic jumps of the B-F type formed on rock cascades. They introduced a new dimensionless parameter with an intention of developing the impacts that are driven by the jump toe

as well as the roller length. This is because they identified that when the chute gradient was variable and the stilling basin slope definitive the changes in the quantity of the variable often led to a fall in the sequent depth ratio. Thus, the ratio of sequent depth decreases, when the value of relative roughness increases for the fixed value of variable.

Valero, Viti, et al., (2018) [65] provided a theoretically elaborate account of prior studies performed in the context of hydraulic jump phenomenon with specific focus on experimental analysis data. They pointed out some ambiguity in the modelling technique due to occurrence of highly turbulent flow features in hydraulic jump flow that cause bubble formation. In their opinion, regarding the scope of the study in the established experimental framework, the assessment of the spread of the liquid phase and the formation of bubbles by PIV and ADV instruments was inadequate. The same would extend to the geometry employed in the numerical modelling where the impacts of scale or inflow-outflow flow rate fluctuations are also manageable.

Eshkou, Dehghani, et al. (2018) [66] investigated the characteristic of hydraulic jump in a gradually varied area of channel provided with series of inclined baffle blocks. They said that baffle blocks offer even better jump characteristics than the open diverging channel in the absence of baffle blocks. Furthermore, they also discovered that the manner in which the baffle blocks are arranged systematically in a converging manner towards the flow direction enhances the properties of jumps. They undertook an experiment of the variety of angles that were tried (10, 20, 30, 40, 50, 60, 70, 80) of which the only convergence angle of 30 degrees was able to factor the top-quality results. In this case the average depth of the jump is reduced by approximately 16% and the average length of the jump reduced by approximately 35% on average.

Felder and Chanson, (2018) [67] investigated the motion of the air–water mixture and some general features of the flow in a hydraulic jump over a rough regular bed with large equally spaced protuberances and smooth bed. They considered Froude numbers between 1 and 10.5 to 6.3 in their study. From these observations, they found that bubble count rate and void fraction near the jump toe increased during the first several minutes of the hydraulic jump when macroroughness was applied. In the second half,

the cross-sectional area of the bed of the water course increases unevenly and up to a part that was as clear as 50 meters downstream strong indications of turbulence were observed.

Salmasi and Samadi, (2018) [68] investigated the flow behaviour, energy loss/dissipation and other hydraulic phenomena occurring over stepped spillways through the Ansys Fluent computational models. They described a step spillway case, where they used the $k-\epsilon$ turbulence model, as well as the Volume of fluid (VOF) model in order to assess the scale of turbulence as well as the associated interface. The discrepancy between two groups of data indicated that the numerical model possessed an acceptable level of capability in foreseeing flow characteristics and energy loss, and most of the potential errors seemed to be falling within 3 to 6%.

Jesudhas, Balachandar, et al. (2018) [69] conducted an ideal a classical hydraulic jump with maximum inlet Froude number of 8.5 using a DES complemented with VOF for the free-surface capturing, allowing a better insight into the dynamics of turbulence in a hydraulic jump. This was important in offering information on the turbulence characteristics and flow structure of the hydraulic jump that lacked representation in prior experiments. They are some features such as the enlargement of the shear layer. They compared their deterministic and stochastic simulation results with the experimental results concerning mean flow profiles and free surface profiles, velocity fields and turbulence characteristics, and got fairly good correlations.

Al-Hashimi, Saeed, et al. (2019) [70] presented the results of in-depth investigation of turbulent characteristics of a classical hydraulic jump created with minimum Froude number of inlet flow is 8.5. To simulate the hydraulic jump, they used an unsteady three-dimensional DES model in combination with the VOF approach for surface capture. They said that in the area of their investigations concerning the wall jet flow and the interaction between the roller area they used such values to compare the simulation data obtained with the experimental data taken. In the course they noticed that the flowing shear layered continues to amplify while implying to the free surface and the free surface is marked by features, which include rupture and very violent oscillations. Their results emphasized the fact that there should exist a mesh resolution

capable of capturing details of the complex bubbly flows solution which is computationally intensive particularly at a higher Froude number.

Kumar, Kumar, et al. (2019) [71] studied the impact of the roughness heights and slopes on the response of hydraulic jump like sequent depth ratio using laboratory and numerical experiments. They applied Artificial neural network (ANN) as a numerical technique. Two positive bed slopes and crushed, rounded aggregates were used to attain needed roughness at the flume bed. At the same flume condition, they noted that crushed aggregates lower the sequent depth ratio by a factor of 35% in contrast with rounded aggregate. However, it was observed that the sequent depth ratio is approximately 45 percent less as the bed slope is increased.

Torkamanzad, Dalir, et al. (2019) [72] discussed how roughness elements and abrupt asymmetrical expansions affected the jump length, sequent depth, energy dissipation and stability of the jump when compared to classical hydraulic jumps in smooth, rectangular channels. They undertook a number of tests in an open and extended channel that included four physical models of different expansion rates. To generate different types of rough bed conditions, they considered two values of heights of element's roughness equal to 1.4 and 2.8 cm. According to the data obtained, the post jump depth and jump length outcomes were decreased with the roughness height increase and the expansion ratio decrease.

Yildiz, Marti, et al. (2020) [73] examined the location of hydraulic jump occurring in a rectangular channel after a sluice gate. The flow conditions were simulated by them using Fluent software. As observed from the simulation they noted that the values arrived at by this method complied with the experimental findings. They also discovered that direction of the jump varies with the discharge, hydraulic geometry of the flume, and position of the step.

Jesudhas, Balachandar, et al. (2020) [74] focused on the results of the study regarding the phenomenon of Symmetric Submerged Spatial Hydraulic Jumps (SSHJ). They employed three-dimensional, unsteady detached eddy simulations coupled with Volume of Fluid multiphase method as a tool for the flow surface modelling. They identified that the jump roller is relatively thicker all across the channel conveyance

but thinner in the middle part of the flume which may lead to increased scour near the walls. Localization highlighted temporal oscillations at the outlet of the conduit. The following are the criteria for identifying vortices; The formation of series of rectangular vortex rings at the exit of the wall jet inside the conduit. These rings got distorted and broken down into the above-said smaller vortex structures existing in the system because of pressure differences in velocities. This made it clear that the prime cause of turbulence in the SSHJ flow field was the dimensional shear layer arising out of the expanding wall jet. The different rollers or structure of beds cause the disintegration of the eddies into little eddies or the increase in the dissipation of energy of turbulence.

Macián-Pérez, Bayón, et al. (2020) [75] performed a numerical analysis on change in energy in case of the hydraulic jump in a rectangular channel. For a particular initial Froude number, they used the standard large scale CFD solvers of Open FOAM and FLOW-3D to study the phenomenon more elaboratively. In order to check the effectiveness of the results to a real building they compared between the physical simulation results to the results got through the simulation software. They addressed themselves with such aspects like the shape of the free surface, length of the roller system, effectiveness of a jump, distribution of velocities and pressure fluctuations. They pointed out that the obtained simulation result is qualitatively close to the experimental result and also acknowledged the fact that there is a quantitative difference in velocity distribution and pressure fluctuation. Their finding thus indicates that CFD codes were perceived to be useful in the design of energy dissipater hydraulic structures. Consequently, the sequent depth values estimated by Open FOAM were fairly accurate regarding their estimation being more than 97% accurate. Similarly, other software failed to show acceptable performance except FLOW-3D which had accuracy above 94.2% for the same factors.

Ghaderi, Dasineh, et al. (2020) [76] performed a numerical study of free and submerged hydraulic jump with and over three types of roughness in an open channel. These shapes were triangular, square and semioval in shape. They used these shapes in form of strips which were provided along the width of the flume. In this research, they employed the RNG $k-\varepsilon$ turbulence model in order to predict the size of turbulence.

They pointed out that among the three types of roughness which have been tested that the triangular roughness improves the jump characteristics more than the other two.

Hien and Duc (2020) [77] investigated the hydraulic characteristics of the Ngan Truoi construction encompassing the spillway, channel chute, stilling basin. They noticed when using several tools that include the computational fluid dynamics and some physical models. They employed RANS and LES turbulence models. From this they were able to deduce that the RANS model was able to provide better results in predicting the water level, velocity and pressure distribution than LES. They emphasised the importance of a model that would define air entrainment in order to have a better compatibility with the numerical model of the rapid flow of water over the structure. Among the works of greatest interest, the mesh size of 0.1 m provides the best fit to the numerical simulations with physical measurements.

Mahtabi, Chaplot, et al. (2020) [78] analysed the classification of hydraulic jump over rough beds employing a novel method based on a decision tree. They employed 581 data sets with 280 drawn from the natural rough bed and 301 from the artificial rough bed. That used to analyse these data, they categorized these into four groups that are A, B C and D based on the energy that was lost. They pointed out that a condition signified by the rough bed had large impact to the stability of the bed shear stress which in turn enhance the lost energy every time the initial Froude number was raised.

Nandi, Das, et al. (2020) [79] provided a set of numerical data that was expected to describe the formation of a hydraulic jump in a rectangular channel in which the bottom slope is low. They also used MacCormack scheme for the solution of differential equations of the Saint-Venant type in one dimension only. They did twenty laboratory experiments in an attempt to tune the numerical model. These experiments entail characterization of the upstream Froude numbers in the range of 2.17 to 7.0 and were conducted for three bed inclinations of 0, 0.02174 and 0.0475. They noted that there was close correlation between the hydraulic jump experimental data and the simulated hydraulic jump profiles. However, when slopes were not horizontal, the location of the jump was farther downstream than position established

in the experiment. This was especially so when the Froude numbers were high as was suggested in the subsequent experiment presented in this study.

Macián-Pérez, Vallés-Morán, et al. (2020) [80] analysed the spatial shape of the free surface together with velocity field of a hydraulic jump in a USBR II stilling basin to improve understanding of energy dissipation in these structures. They conducted an experimental campaign using innovative tools like a pitot tube and a time-of-flight camera to measure the free surface profile and the general velocity field of the flow. They discussed the strengths and limitations of the experimental tools. The LIDAR provided accurate free surface measurements despite limitations like splashing, while the Pitot tube had difficulties capturing velocities in highly aerated areas. They demonstrated that the maximum velocity within the hydraulic jump is affected by the energy dissipation devices, which cause the maximum velocity to occur near the channel bottom, unlike classical hydraulic jumps. Finally concluded that the hydraulic jump in the stilling basin was found to have a lower sequent depth ratio and higher energy dissipation efficiency compared to classical jumps.

Nikmehr and Aminpour (2020) [81] applied a 2-phase flow theory, utilizing the FLOW-3D software. They used the VOF (Volume of Fluid) method and RNG turbulence models to simulate hydraulic jumps over rough beds in a rectangular flume. They reported that rough beds significantly affect hydraulic jump characteristics by reducing sequent depth and jump length, and by modifying the velocity distribution. They compared the numerical data against experimental data to verify accuracy, particularly focusing on the water surface profile and discharge measurements and reported that numerically derived hydraulic jump length and roller length closely matched experimental data, with mean errors of 4.8% and 10.85% respectively. Therefore, numerical models like FLOW-3D are effective for simulating hydraulic jumps.

Wiest, Júnior, et al. (2020) [82] proposed a procedure for initiating an advancing hydraulic jump with a sloping front at specific points on a Creager spillway. They performed experiments using a two-dimensional model of a hydroelectric power plant's spillway and stilling basin. Their experimental results allowed them to develop

an equation that accurately predicts the start position of the hydraulic jump. That equation was validated through experimental data and comparisons with previous studies. They found that the jump's position is primarily influenced by the incident Froude number and the submergence factor.

Hafnaoui and Debabeche (2021) [83] evaluated Iber software's effectiveness in simulating the location and movement of hydraulic jumps in rectangular channels. They focused on different Froude numbers and sill heights to observe the hydraulic jump's behaviour in rectangular channels. Their simulations closely matched experimental results, particularly for certain Froude numbers. Therefore, they found Iber software valuable for flood risk management and protecting hydraulic structures from erosion.

Gualtieri and Chanson (2021) [84] discussed the complexities of modelling air-water interactions and the need for ongoing improvements in both physical and numerical approaches, including further validation and overcoming scale effects. They considered two case studies to demonstrate these modelling approaches. They stated that laboratory-based physical models have been traditionally used to study self-aerated flows. However, they face limitations, such as scale effects, that hinder accurate extrapolation to real-world systems. Whereas, new development in Computational Fluid Dynamics (CFD) has made it possible to examine such flows in details. While CFD can model full-scale scenarios, it requires proper validation against high-quality experimental data to ensure accuracy.

Maleki and Fiorotto (2021) [85] presented a novel method to define hydraulic jump characteristics over rough beds taking into account both the Froude number and bed roughness. They found that approximately 35% reduction in hydraulic jump length when boulders or gravel protect the bed, minimizing erosion and cavitation. Their analysis reveals that the integrated bed shear stress depends on the Froude number.

Nasrabadi, Mehri, et al. (2021) [86] predicted submerged hydraulic jump (SHJ) characteristics using machine learning methods. They performed experiments in a rectangular flume measuring 9m long, 0.5m wide, and 0.45m deep with minimum initial Froude numbers of 3.5 and maximum of 11.5, and minimum submergence ratios

of 0.1 and maximum of 4. On submerged hydraulic jump characteristics, they used the method called the Developed Group Method of Data Handling and compared with the Group Method of Data Handling model. They trained and tested both models with experimental data to predict relative submergence depth, jump length, and relative energy loss. They reported that the DGMDH model, combining regression and intelligent neural networks, provided more accurate predictions than the GMDH model. Hence, they recommend using DGMDH to estimate submerged hydraulic jump characteristics in hydraulic engineering.

Turker and Valyrakis (2021) [87] intended to evaluate and measure such effects scientifically in terms of the channel bed roughness on hydraulic jumps. They introduced new definitions and models, particularly for the shear force coefficient and roller length, validating their findings through experiments. They conducted 60 experiments using a flume, roughening the channel bottom with different sizes of sand and gravel to observe how various bed roughness levels influence hydraulic jump characteristics. They found that as bed roughness and flow turbulence increase, the length of the hydraulic jump reduces, with the shear force coefficient directly related to bed roughness. They also noted that for Froude number between 1.1 and 9.8, both shear force coefficient and roller length estimated fairly close to the experimental values.

Ghaderi, Dasineh, et al. (2021) [18] conducted numerical simulations to investigate the flow properties of the submerged jump over macrorough beds specifically in triangular strip macroroughness. They used the FLOW-3D model to explore several parameters, including streamlines and flow patterns, velocity profiles, energy loss, turbulent kinetic energy (TKE), shear Stress. They reported that maximum flow velocity occurs closer to the bed in submerged jumps compared to free jumps over smooth and rough beds. Further, distance between triangular macroroughnesses significantly influences the flow behaviour and triangular macroroughnesses increase bed shear stress and energy loss more than smooth beds. They validated the simulation findings against experimental results with a maximum error of 8.41% and a mean relative error of 4.83%.

Hassanpour, Dalir, et al. (2021) [88] provided an experimental study of pressure variation in spatial hydraulic jump that develops in stilling basins with different expansion ratio. They conducted experiments in a rectangular flume with varying expansion ratios i.e., 0.4, 0.6, 0.8, and 1 to simulate hydraulic jumps. They used pressure transducers to measure pressure fluctuations at 31 points along the centreline of the stilling basin. They observed that pressure fluctuations increased significantly near the hydraulic jump toe and then decayed downstream. The Froude number strongly influenced pressure fluctuations, with larger Froude numbers leading to higher pressure fluctuations. The maximum root means square values vary with different expansion ratios, with the highest observed at an expansion ratio of 0.8.

Dasineh, Ghaderi, et al. (2021) [89] predicted the properties of jumps on triangular strip rough bed using both soft computing methods and numerical modelling. They utilized Computational Fluid Dynamics (CFD) via the FLOW-3D software to simulate hydraulic jumps on triangular bed roughness under various hydraulic conditions. They reported that the FLOW-3D® model showed a mean relative error of 4.1%, indicating a strong predictive capability. To verify the effectiveness of the CFD model's, they employed three artificial intelligence methods such as Support Vector Machines, Gene Expression Programming, and Random Forest. Among the soft computing methods, SVM was identified as the most accurate, with the lowest root mean square error and highest correlation coefficient. They concluded that hydraulic jumps on rough beds exhibited smaller sequent depths and shorter lengths compared to smooth beds, leading to greater energy dissipation.

Wuthrich, Shi, et al. (2022) [90] presented hydraulic jumps with low inflow Froude numbers i.e., 2.1 and 2.4, exploring their air-water flow properties under high Reynolds number conditions. They involved experimental tests using dual-tip phase-detection probes and high-speed video cameras for capturing characteristics features such hydraulic jump surface patterns, voids percentage, bubble formation rate, and interfacial velocities. They showed that data from the sidewalls might not accurately reflect the behaviour at the centre, especially regarding air entrainment and flow dynamics, as sidewalls tend to affect flow properties differently. They concluded that hydraulic jumps with low Froude numbers were found to be effective energy

dissipators, contributing valuable insights to the design of hydraulic structures for long-term operation.

Parsamehr, Kuriqi, et al. (2022) [91] conducted a comprehensive experimental investigation on hydraulic jumps occurring downstream of hydraulic structures. They tested various roughness elements on horizontal and adverse bed slope of -1.5% and -2.5% to study their effects on jump properties, such as energy dissipation, post jump depth ratio, and roller length. They found that both increased slope steepness and roughness height enhanced energy dissipation by forming larger size eddies and huge turbulence, significantly reducing the sequent depth ratio. The energy loss with roughness elements ratio of 1.33 on a -2.5% slope was 19.6% higher than that of a classical hydraulic jump. Also, the bed shear coefficient increased by 12.4 times compared to a smooth bed on a -2.5% slope.

Bahmanpouri, Gualtieri, et al. (2023) [92] examined the effects of bed roughness on turbulence properties like intensity, correlation time scales, and advective length scales in a hydraulic jump. They conducted experiments in a rectangular flume where natural river pebbles were placed on the bed for roughness. They captured air-water flow data using phase-detection probes under different hydraulic jump conditions. They reported that turbulence intensity and the presence of larger eddies were greater on the rough bed, indicating more complex interactions between the air-water flow and bed roughness.

Zaffar and Hassan (2023) [93] used FLOW-3D models to simulate the flow behaviour and hydraulic jumps in wedge-shaped baffle blocks stilling basin. They focused on variables such as ambient surface profiles, length of roller, longitudinal velocity profiles, and turbulent kinetic energies. They reported that the new wedge-shaped baffle blocks stilling basin stabilized the hydraulic jump and shortened the jump lengths compared to traditional USBR-Type III basins. They concluded that the FLOW-3D software successfully simulated and validated the hydraulic behaviour for different stilling basin designs, providing reliable insights into the performance of the new wedge-shaped splitter blocks.

Macián-Pérez, García-Bartual, et al. (2023) [94] investigated the efficiency of stilling basin and more precisely this type of stilling basins that include an abrupt change in the depth through modelling with the help of CFD methods. They analysed six discrete numerical models, each of which they explored with two imaginable relative heights of an abrupt drop. They studied the effect of a decrease in the energy dissipation rate and changes with regard to the dimensions and hydraulic characteristics of the jump. They also looked into the pressures acting on the streambed, and the variation in the pressures. They indicated that the sharp decrease causes low subcritical depths and consequently small basin dimensions and somewhat longer hydraulic jump length. As mentioned earlier it was to the advantage if there was a sudden drop when seeking to control hydraulic jumps in the stilling basin.

CHAPTER 3: METHODOLOGY

3.1 General

This chapter describes the method that has been used to study the free and submerged hydraulic jumps in a prismatic rectangular channel on smooth and macrorough beds. Figure 3.1 depicts the definition sketch of free and submerged hydraulic jump, along with their associated parameters.

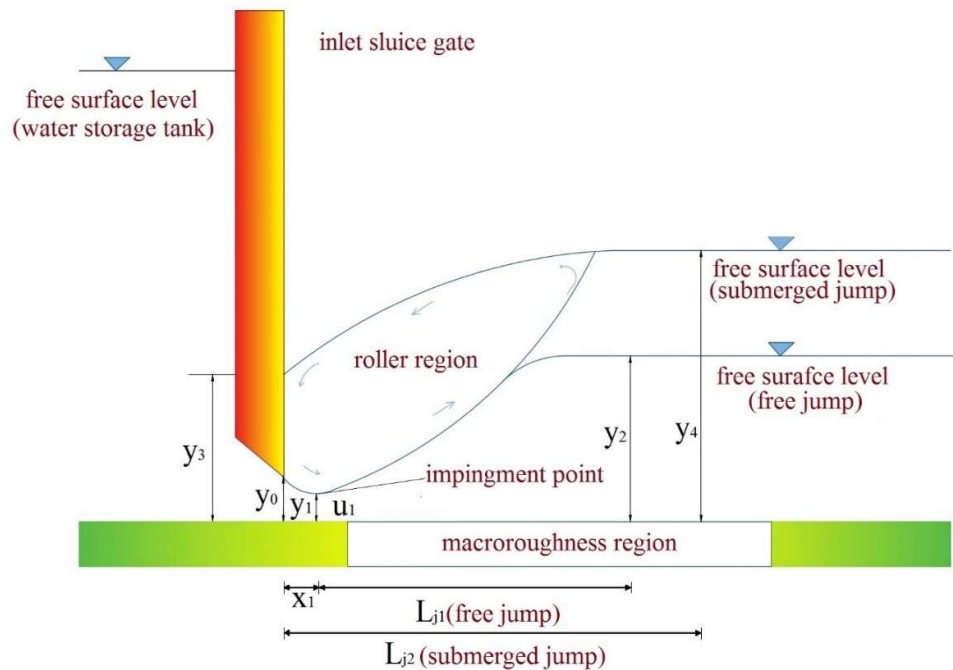


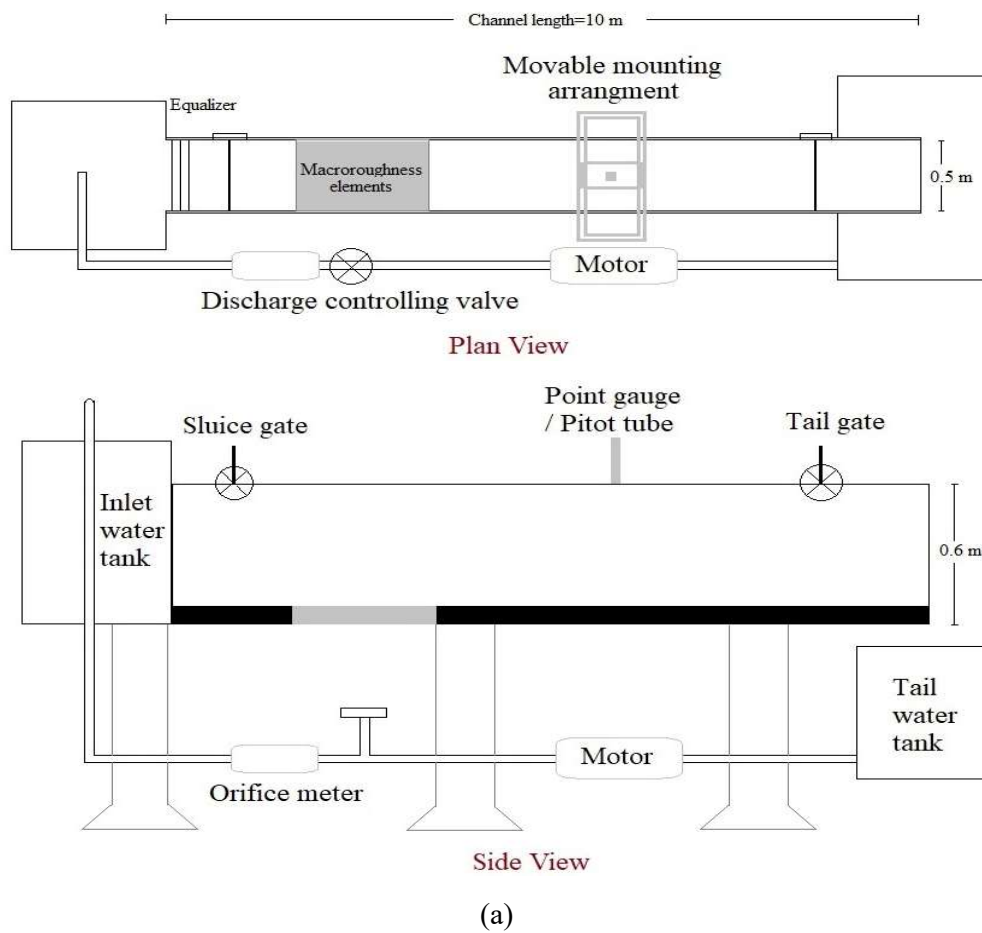
Figure 3.1 Definition sketch of free and submerged hydraulic jump showing relevant parameters

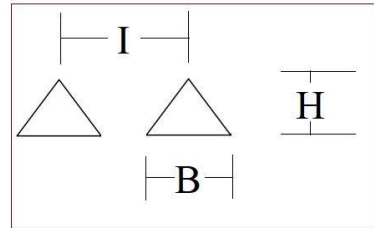
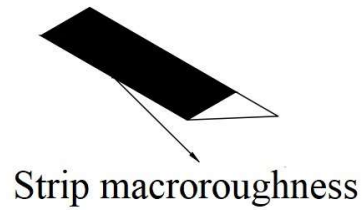
The research work was conducted using numerical simulation and subsequently validated against the physical findings of the present study. It should be noted that the validation was performed for only three specific cases. The initial phase involved conducting physical experimental analysis of hydraulic jump on a smooth bed and two distinct types of artificial macrorough beds, namely triangular and trapezoidal, within a laboratory flume. In addition, numerical simulations were performed using Ansys Fluent R22 software, taking into account the experimental data. The findings of the

numerical simulation were compared with experimental findings and previous studies for only three cases. The following sections outline the detailed steps of the experimental and numerical analysis.

3.2 Physical experimental model

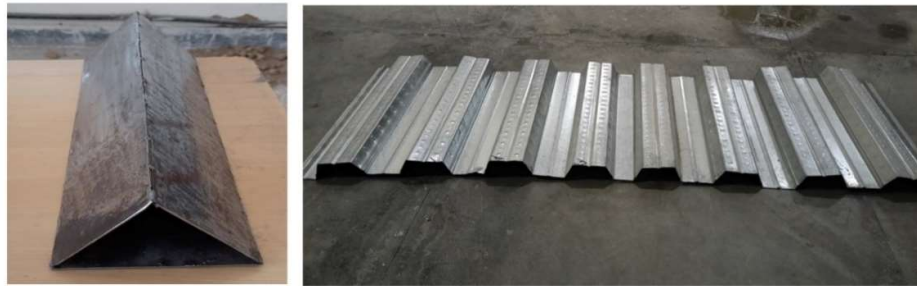
In the present work, hydraulic jump phenomenon was studied in an open channel prismatic rectangular flume at Delhi Technological University's hydraulics lab which was 10.0 m long, 0.5 m deep, and 0.5 m wide.





$$H=0.055 \text{ m}, B=0.11 \text{ m}, I=0.22 \text{ m}$$

(b)



(c)

Figure 3.2 (a) Definition sketch of experimental setup; (b) Definition sketch of strip macroroughness; (c) Physical strip macroroughness elements

Figure 3.2 represents the experimental setup and physical macroroughness prepared with iron material that were positioned across the whole width of the flume up to certain length to create bed macroroughness. For every round of the experiment, the bed macroroughness length was kept constant. On a mild bed slope with varying Fr_1 numbers ranging from 3.5 to 7.0, forty-two runs were conducted. Discharge and velocity were measured with the help of digital discharge meter and digital pitot tube (L-type pitot tube of 6 mm diameter and digital manometer with an accuracy of $\pm 0.5\%$ full scale deflection), respectively. Other relevant measurements such as water depth were done with the help of rail mounted pointer gauges with an accuracy of 0.2 mm. Figure 3.2 represents the experimental setup and physical macroroughness used in

experiments. Table 3.1 depicts the parametric variations considered in the present study.

Table 3.1 Parametric variations of flow for present study (numerical and experimental)

<i>Bed Type</i>	<i>Q (l/s)</i>	<i>y_o (cm)</i>	<i>y_l (cm)</i>	<i>u (m/s)</i>	<i>Fr_l</i>
Smooth	15-25	2.5-3.0	1.1-1.5	1.24-2.23	3.7-7.0
Rough	15-25	2.5-3.0	1.1-1.5	1.24-2.23	3.7-7.0

3.3 Numerical Simulation

The objective of this study was to employ Computational Fluid Dynamics (CFD) to analyses the hydraulic jump phenomenon in a rectangular open channel, both over smooth and rough beds. The investigation was conducted numerically. Figure 3.3 illustrates the standard procedure employed in numerical simulation.

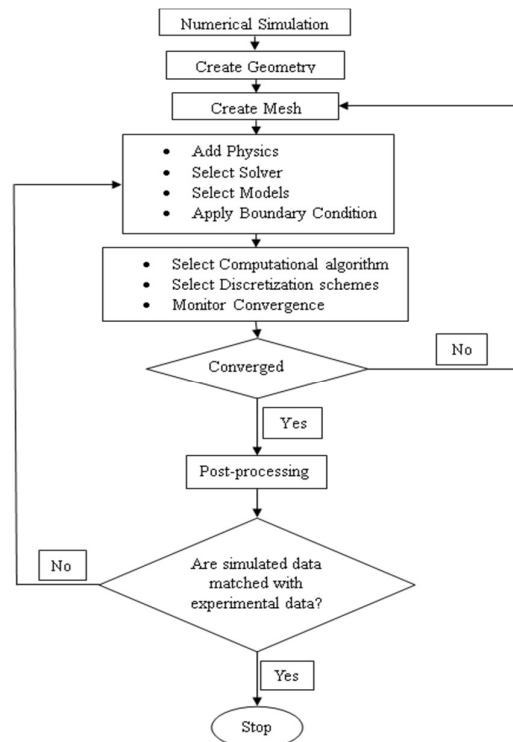


Figure 3.3 Flow chart of adopted methodology for numerical simulation

3.3.1 Governing Flow Equations

The Navier-Stokes equations were historically identified as the momentum equations for a viscous flow. Claude-Louis Navier (1838) [95] first derived the equations from Newtonian mechanics of fluids taking into consideration the forces because of viscosity. However, in the modern context of Computational Fluid Dynamics (CFD) these terms are used to refer to the entire set of flow equations required to be solved for viscous flow. That is as it pertains to continuity, energy, and momentum relations respectively. Consequently, in the CFD literature the phrase numerical solution to the Navier-Stokes equations is generally understood to mean the numerical solution of the full system of equations. Thus, in the context of CFD literature, ‘Navier-Stokes solution’ means the solution to the problem of viscous flow wherein the full set of governing equations has been employed.

3.3.2 Navier Stokes Equation

CFD simulation of the fluid flow is based on the Navier-Stokes equations as a primary navigation of the system. Viscous fluid substances can be defined and explained by the set of partial differential equations for their motion. These equations are obtained when applying Newton second law on the fluid flow and considering the forces of viscosity and pressure forces in the fluid.

The continuity equation is as follows;

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \vec{v}) = 0 \quad (10)$$

The momentum equation is as follows:

$$\frac{\partial}{\partial t}(\rho \vec{v}) + \nabla \cdot (\rho \vec{v} \vec{v}) = -\nabla p + \nabla \cdot (\vec{\tau}) + \rho \vec{g} + \vec{F} \quad (11)$$

Where, ρ is the fluid density, \vec{v} is the velocity vector, p is the static pressure, $\vec{\tau}$ is stress tensor, $\rho \vec{g}$ is the gravitational body force, \vec{F} is the external body force.

Stress tensor, $\vec{\tau}$ is given by

$$\vec{\tau} = \mu \left[(\nabla \vec{v} + \nabla \vec{v}^T) - \frac{2}{3} \nabla \cdot \vec{v} I \right] \quad (12)$$

3.3.3 Volume of Fluid (VOF) Model

The VOF model is utilized to monitor the volume fraction and even predict the extent of each fluid contained in each computational cell. The most important goal to achieve is to accurately represent the geometry and behaviour of the interface between two nonmixing fluids. The method is very efficient for the problems which relate to the interaction between the phases, for example, the flows of free surfaces and the impact of surface tension forces.

Earlier works of Bayon-Barrachina and Lopez-Jimenez (2015) [55] & Ghaderi et al. (2021) [18] used computational fluid dynamics (CFD) technique to track the free surface of the flow. This was made possible through a complicated method known as the volume of fluid (VOF) model to make the animations. In this study, the VOF model has been used as it helps in tracking the volumetric ratio of interface in two-phase gas/liquid applications. In this case, water is a liquid while air on the other hand is a gas. On this basis, the physical classification of matter is made.

Canonsburg (2013 [24] discussed the basic definition and governing equations of Volume of Fluid (VOF) model. Model employs a volume fraction α_q to represent the fraction of a cell that is filled with the q^{th} fluid. In a two-phase system, the volume fraction α_q of fluid 1, such as water, is defined as follows: In a two-phase system, the volume fraction α_q of fluid 1, such as water, is defined as follows:

- The value of $\alpha_q=0$ is relevant to that cell in which there is only the second type of fluid, for instance, air.
- $\alpha_q=1$ corresponds to a cell containing only fluid 1.
- $0<\alpha_q<1$ corresponds to a cell containing the interface between the two fluids.

Volume Fraction Equation:

Transitions between the phases occur based on the solution to an equation of continuity associated with the volume fraction of one or more phases. Performance of the model for the q^{th} phase can be described by the following equation:

The equation for the q^{th} phase takes the following form:

$$\frac{1}{\rho_q} \left[\frac{\partial}{\partial t} (\alpha_q \rho_q) + \nabla \cdot (\alpha_q \rho_q \mathbf{v}_q) \right] = S_{\alpha q} + \sum_{p=1}^n (\dot{m}_{pq} - \dot{m}_{qp}) \quad (13)$$

Where \dot{m}_{pq} is the mass transfer from phase q to phase p and \dot{m}_{qp} is the phase p to phase q. The source term $S_{\alpha q}$ is zero by default.

The equation of volume fraction of the primary phase will not be solved for this case. However, the volume fraction of the primary phases will be estimated a priori according to the following constraint:

$$\sum_{q=1}^n \alpha_q = 1 \quad (14)$$

Volume fraction equation may be solved either numerically explicitly incorporating time as a variable or using it for iterative computations.

Implicit Time Discretization:

It should be mentioned that in employing the implicit methods the future state of the system is found by solving a system of equations. These methods are usually stable, thereby allowing more forecasts to be made using larger steps than are possible with explicit methods. However, these tasks are computationally more intensive mainly because of the importance given to iterative solvers.

Explicit Time Discretization:

In explicit time discretization one determines the state of the system at the next time step merely utilizing the information of the current time step. What is more, the implementation of this method is usually easier, and the computational complexity per time step is less. However, to have a stable solution, it is required to take small time steps especially in the problems, which involve high velocities and large gradients.

That is, whether one uses explicit or implicit time discretization depends on the specific requirement of the problem as it pertains to stability, accuracy and computational expense. It is suggested to employ explicit methods if the size of time steps is small because the methods are rather simple and fast. However, implicit

methods are more appropriate for use when larger time steps are needed or the system of equations is stiff.

Momentum Equation:

The momentum equation of the VOF model is for the whole mixture and includes contributions coming from both phases:

$$\frac{\partial(\rho\vec{u})}{\partial t} + \nabla \cdot (\rho\vec{u}\vec{u}) = -\nabla p + \nabla \cdot \tau + F \quad (15)$$

where:

- ρ is the density of the mixture, calculated as $\rho = \alpha\rho_1 + (1 - \alpha)\rho_2$, where ρ_1 and ρ_2 are the densities of fluids 1 and 2, respectively.
- p is the pressure.
- τ is the stress tensor, incorporating the effects of viscosity, calculated as $\tau = \mu(\nabla\vec{u} + \nabla\vec{u}^T)$, where μ is the dynamic viscosity of the mixture.
- F includes body forces such as gravity and surface tension.

Energy Equation:

The energy equation, also shared among the phases, is shown below;

$$\frac{\partial}{\partial t}(\rho E) + \nabla \cdot (\vec{u}(\rho E + p)) = \nabla(k_{eff}\nabla T) + S_h \quad (16)$$

The VOF model treats energy, E , and temperature, T , as a mass-averaged variables.

The parameters, ρ and k_{eff} are all per phase. The source term S_h which resides on the right-hand side includes the effects due to radiation as well as any other volumetric heat source, as well as any other volumetric heat source.

The simulation of multiphase flows in Ansys Fluent requires careful consideration of surface tension and wall adhesion. These parameters are particularly important when dealing with immiscible fluids. For the water-air interface, the Surface Tension Coefficient was set to 0.072 N/m. To accurately model the effect of open channel flow,

the Open channel sub-VOF model was selected. The above model covers for the situation that arises due to the existence of free surface between the fluid in motion and the fluid on top of the former.

3.3.4 Turbulence models

To study the hydraulic jump characteristics using CFD it has become imperative to simulate the flow using right techniques. That is why RANS equation is obtained from the Navier-Stokes equation. Here the velocity and pressure fields are described simply as the meantime and adding a fluctuating term (Gualtieri and Chanson 2021) [84].

Turbulence modelling is the most demanding tool in Fluent software for the simulation of fluids and it is used particularly when the turbulent flow is disturbed and chaotic. There are two commonly used turbulence models: These models are split into the zero-equation model and the two equation models. Zero equation model is regarded to be the simplest form of the turbulence modelling as a matter of fact. It is known by the names of Algebra of Mixing, Algebraic Model, or Mixing Length Model. This model is based on log laws and can be used for the simplest and highly deterministic turbulent, thus it is applied when the nature of the flow is rather evident. There is no other transport equations solved for turbulence quantities. Two Equation Models are well acclaimed and widely used for variety of turbulent flows. These models incorporate two extra transport equations to consider the impact of turbulence: There are common variables such as the turbulence kinetic energy (k) and another for the turbulence dissipation rate (ϵ) or specific turbulence dissipation rate (ω). Ansys Fluent offers two commonly used two-equation models: two models, of which the first is known as the k - ϵ model and the second as the k - ω model.

Lu et al. (2022) [96] in their study devoted to the sensitivity analysis of the mesh and turbulence using the Ansys Fluent software. They concentrated on a nuclear fuel bundle in a standard pressure flume and explored how the decision of the meshing technique and the turbulence models can influence the results of velocity profile predictions. Such factors are known to have a very close and direct relationship to such outcomes and their discoveries supported this fact.

According to Bayon-Barrachina and Lopez-Jimenez (2015) [55], the RNG k- ε model gives better result in analyzing hydraulic jumps than other models consisting of SST k- ε and standard k- ε . Nikmehr and Aminpour (2020) [81] checked the reliability of the RNG k- ε model by conducting numerical simulation of hydraulic jumps over trapezoidal rough beds. Further numerical analyses of hydraulic jumps in macrorough bed conditions carried out by Ghaderi et al. (2021) [18] suggest that the RNG k- ε model is rather similar to the standard k- ε model, although there are some differences. Canonsburg (2013) [24] suggest that this model provides better modelling of whirling. The present study compared standard k- ε , RNG k- ε , Realizable k- ε , and SST k- ω turbulence models to get the best turbulence model.

Transport Equation for the Standard k- ε model:

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_i} \right] + G_k + G_b - \rho \varepsilon - Y_M + S_k \quad (17)$$

$$\begin{aligned} \frac{\partial}{\partial t}(\rho \varepsilon) + \frac{\partial}{\partial x_i}(\rho \varepsilon u_i) \\ = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_i} \right] + C_{1\varepsilon} \frac{\varepsilon}{k} (G_k + C_{3\varepsilon} G_b) - C_{2\varepsilon} \rho \frac{\varepsilon^2}{k} + S_\varepsilon \end{aligned} \quad (18)$$

$$\mu_t = C_\mu \frac{\rho k^2}{\varepsilon} \quad (19)$$

$C_\mu, C_{1\varepsilon}, C_{2\varepsilon}, \sigma_k, \sigma_\varepsilon$ are the model parameters of RNG k- ε turbulence model; these constants have following values: 0.09, 1.44, 1.92, 1.0 & 1.3, respectively.

Transport Equation for the RNG k- ε model:

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j} \left[\alpha_k \mu_{eff} \frac{\partial k}{\partial x_i} \right] + G_k + G_b - \rho \varepsilon - Y_M + S_k \quad (20)$$

$$\begin{aligned}
\frac{\partial}{\partial t}(\rho\varepsilon) + \frac{\partial}{\partial x_i}(\rho\varepsilon u_i) \\
= \frac{\partial}{\partial x_j} \left[\alpha_k \mu_{eff} \frac{\partial \varepsilon}{\partial x_i} \right] + C_{1\varepsilon} \frac{\varepsilon}{k} (G_k + C_{3\varepsilon} G_b) - C_{2\varepsilon} \rho \frac{\varepsilon^2}{k} - R_\varepsilon \\
+ S_\varepsilon
\end{aligned} \quad (21)$$

$$\mu_t = C_\mu \frac{\rho k^2}{\varepsilon} \quad (22)$$

Those symbols basically refer to the RNG k - ε turbulence model of the present CFD model, and the exact values associated with the model parameters $C_\mu, C_{1\varepsilon}, C_{2\varepsilon}, \sigma_k, \sigma_\varepsilon$ are 0.845, 1.42, 1.68, 1.0 & 1.3, respectively.

Transport Equation for the Realizable k - ε model:

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_i} \right] + G_k + G_b - \rho \varepsilon - Y_M + S_k \quad (23)$$

$$\begin{aligned}
\frac{\partial}{\partial t}(\rho\varepsilon) + \frac{\partial}{\partial x_j}(\rho\varepsilon u_j) \\
= \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_i} \right] + \rho C_{1\varepsilon} S_\varepsilon - \rho C_{2\varepsilon} \frac{\varepsilon^2}{k + \sqrt{\nu \varepsilon}} \\
+ C_{1\varepsilon} \frac{\varepsilon}{k} C_{3\varepsilon} G_b + S_\varepsilon
\end{aligned} \quad (24)$$

Where;

$$C_1 = \max \left[0.43, \frac{\eta}{\eta + 5} \right], \quad \eta = S \frac{k}{\varepsilon}, \quad S = \sqrt{2 S_{ij} S_{ij}}, \quad \mu_t = C_\mu \frac{\rho k^2}{\varepsilon}$$

The difference between the realizable, standard and RNG k - ε models is that C_μ is not constant any longer; rather it is calculated using

$$C_\mu = \frac{1}{A_o + A_s \frac{k U^*}{\varepsilon}} \quad (25)$$

$C_{1\varepsilon}, C_2, \sigma_k, \sigma_\varepsilon$ are the model parameters of realizable k- ε turbulence model; these constants have following values: 1.44, 1.9, 1.0 & 1.2, respectively.

k: The Turbulent kinetic energy is degree of energy related to the turbulent velocity fluctuations in the given fluid flow. This measures or quantifies the nature of the turbulence in the given flow of the fluid.

ε : Proportion of the energy contained within the turbulence to the rate at which the energy from turbulence is dissipated. This is the rate at which energy of turbulence is dissipated as heat, because of viscosity.

t: time variable.

ρ : density of the fluid.

x_i : coordinate in the i-axis of a Cartesian reference system.

μ : dynamic viscosity of the fluid.

μ_t : Turbulent eddy dynamic viscosity literally refers to the ability of the turbulent eddy movement to result to higher dynamic viscosity. It is an effective model parameter which describes turbulence effects or more strictly the effective viscosity connected with them.

G_k : the technical term for the kinetic energy of the turbulence generally used in wind power production. It is the provision of the rate of production of turbulent kinetic energy stemming from the active extrapolation of turbulent stresses.

G_b : buoyancy effect term. It is the reflection of the effect of Buoyancy forces on the modelling of turbulence of the flow.

Y_M : dilatation oscillation effect term. This applies how the variation in forms of the elements of fluids affects the turbulence.

S_k and S_ε : these terms are logarithmic of 'k' and ' ε '. These terms provide further details of the variation in the rate of change in k and ε in the model of turbulence.

Transport Equation for the SST k - ω model:

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j} \left(\left(\mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right) + \tilde{G}_k - Y_k + S_k \quad (26)$$

$$\frac{\partial}{\partial t}(\rho \omega) + \frac{\partial}{\partial x_i}(\rho \omega u_i) = \frac{\partial}{\partial x_j} \left(\left(\mu + \frac{\mu_t}{\sigma_\omega} \right) \frac{\partial \omega}{\partial x_j} \right) + G_\omega - Y_\omega + D_\omega + S_\omega \quad (27)$$

Where;

$$\mu_t = \frac{\rho k}{\omega} \frac{1}{\max \left[\frac{1}{\alpha^*}, \frac{SF_2}{\alpha_1 \omega} \right]}, \quad \tilde{G}_k = \min (G_k, 10\beta^* \rho k \omega), \quad Y_k = \beta^* \rho k \omega,$$

$$D_\omega = 2(1 - F_1) \frac{\rho \sigma \omega_2}{\omega} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j}$$

\tilde{G}_k : stands for production of turbulence kinetic energy by mean velocity gradients

G_ω : represents the generation of ω

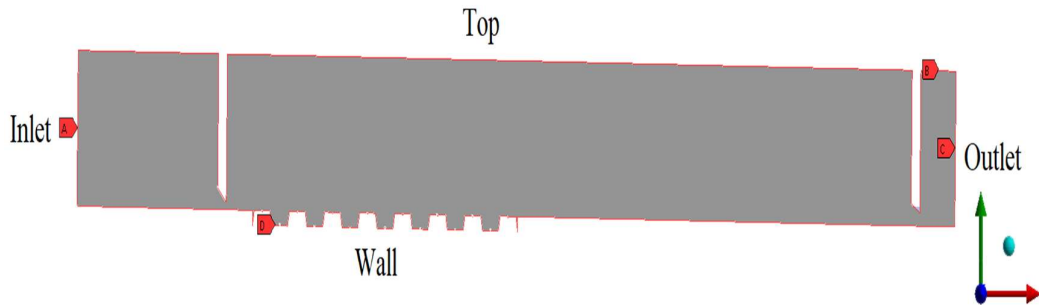
S_k & S_ω : are user defined source term

D_ω : represents the cross-diffusion term

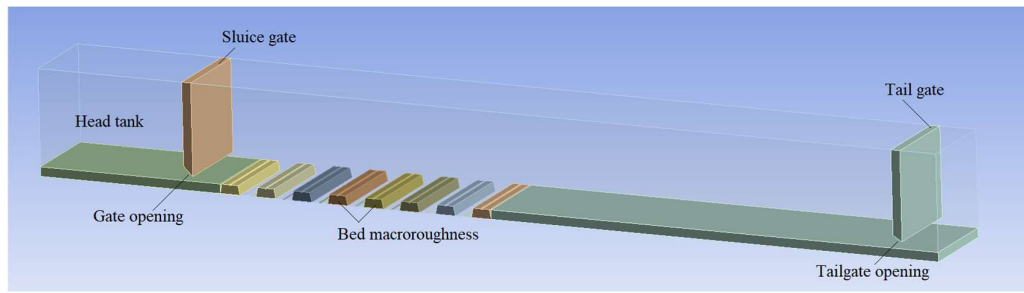
F_1 is a blending function

3.4 Geometry

Figure 3.4 illustrates the overall geometry of the numerical simulation, which was constructed using Ansys Design Modular to match the dimensions of the experimental flume. The position of the sluice gate was adjusted to 0.87 m in the direction of flow.



(a)



(b)

Figure 3.4 Model geometry of simulation setup showing relevant parameters associated with the flume (a) 2-dimensional; (b) 3-dimensional

3.5 Meshing

The Ansys Fluent software requires a finite volume mesh to be present in the computational domain for accurate calculations Canonsburg [24]. Two crucial aspects must be taken into account when creating the mesh in the computational domain. It is important to ensure that the mesh is sufficiently refined to accurately capture the rapid spatial variations of velocity or other significant flow characteristics, such as temperature. These pertain especially to parts close to a wall or a solid formation in the flow path especially when in turbulent flow regime where the possibility of significant changes in the flow across a short distance is rather high. Hydraulic jump simulation is the main focus of enquiry in this study. Due to the instability of hydraulic jump, it obviously requires a more number of cells or fine mesh to obtain a high accuracy and reliability of the simulation results. One disadvantage of having a higher number of cells in a simulation is that the more cells there are the longer it will take to compute and the greater the amount charged. Additionally, it is important to acknowledge that dealing with a higher number of cells can make the post processing of the results more challenging.

Figure 3.5 demonstrates the mesh surrounding the geometry. A mesh analysis was conducted in a meshing setup, where the different faces were assigned suitable names. Additionally, inflation layers were incorporated at the flume's bed to enhance the flow behaviour.

3.5.1 Mesh sensitivity analysis

Mesh sensitivity analysis is a statistical technique used to evaluate the impact of parameter or input changes on a model's output. In order to achieve accurate results while keeping computational costs at a minimum, it is necessary to perform sensitivity analysis during the numerical simulation. This analysis helps determine the ideal combination of mesh and turbulence for the simulation.

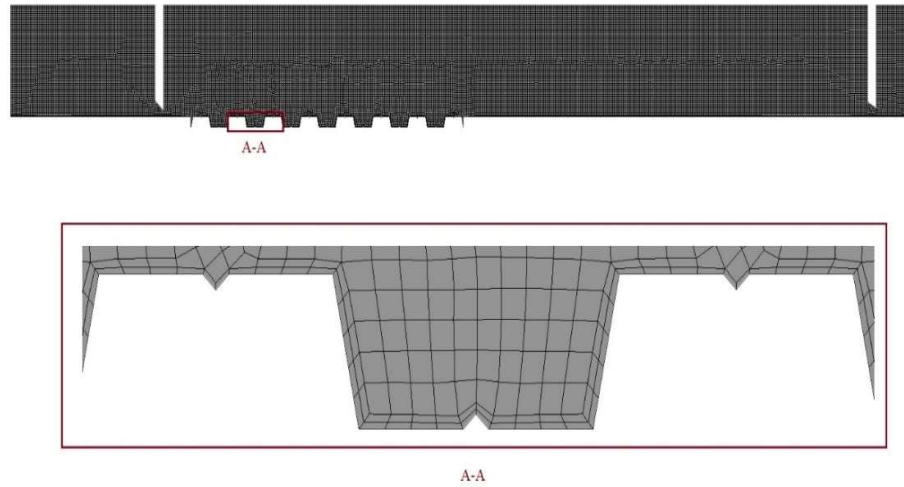


Figure 3.5 Mesh structure allocated to model geometry

This study conducted a sensitivity analysis on grid independency and turbulence models. This study examined the sequent depth ratio (y_2/y_1) as a key parameter for conducting a sensitivity analysis on the grid and turbulence model. The approach used in this study was similar to that of previous research conducted by Bayon-Barrachina and Lopez-Jimenez (2015) [55] and Dasineh et al. (2021) [89]. For the sensitivity analysis six different grid sizes ranging from coarse to fine were considered. Figure 3.6 illustrates the fluctuation of the relative error in predicting the sequent depth ratio across various turbulence models and mesh element sizes. The various models demonstrate minimal variation in their predictions of sequent depth ratio. The RNG k- ϵ model produced the best results with the least amount of error, followed by the realizable k- ϵ , SST k- ω , and standard k- ϵ models. Figure 3.6 demonstrates that as the mesh element size decreases, the relative error in predicting the sequent depth ratio

also decreases. After reaching a mesh element size of 9 mm, the error remains constant, indicating that further reduction in size is not necessary to avoid excessive computational cost. The reported maximum error at a size of 9 mm with RNG k- ϵ was less than 2%.

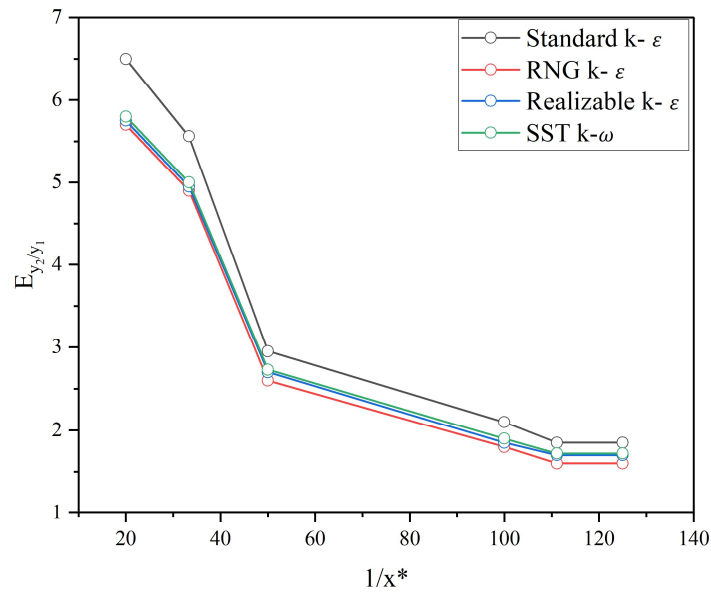


Figure 3.6 Mesh and turbulence model sensitivity analysis on y_2/y_1 prediction

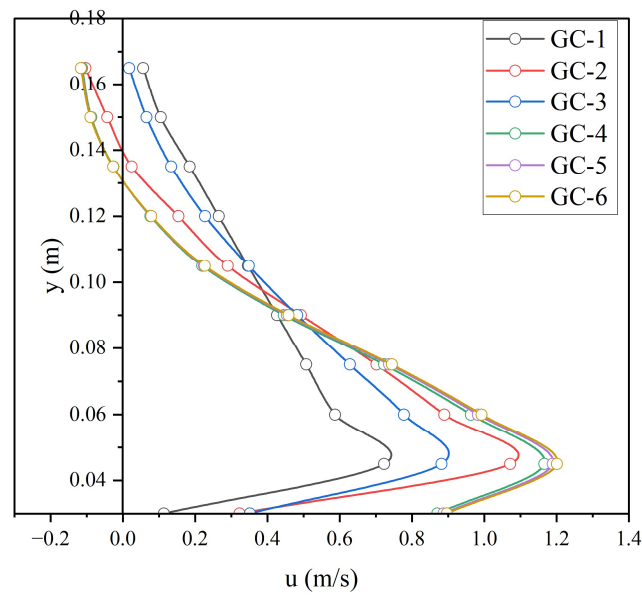


Figure 3.7 Sensitivity analysis of mesh size for horizontal component of velocity profile

In addition, Figure 3.7 illustrates the results of a mesh sensitivity analysis on the longitudinal velocity profile of a free hydraulic jump on a smooth bed. The analysis was conducted using the RNG k- ϵ turbulence model, following the procedure outlined by Celik et al. (2008) [96]. From Figure 3.7 it is evident that the fluctuation in the longitudinal velocity profile is minimal for GC-5 & GC-6. Therefore, there is no need for further reduction in mesh size to save computational costs. Therefore, the present study considered the 9 mm mesh size and RNG k- ϵ turbulence model to analyse the hydraulic jump over smooth and macrorough beds using cost-effective numerical methods.

3.6 Boundary conditions

Specifically for hydraulic jump replication and especially when trying to obtain highly accurate results, the boundary conditions should be selected properly. Typically, boundary conditions address inlet, outlet, wall and atmosphere surfaces.

3.6.1 Inlet and outlet

The pressure inlet has been selected to create a sudden increase near the flume, while gauge pressure has been chosen for the top surface and outflow of the flume. The initial settings at the inlet and outflow are adjusted to replicate the conditions found in experimental scenarios. However, Chapter 4 discussed the various boundary conditions to conduct the numerical simulations of hydraulic jump.

3.6.2 Near wall treatment

Wall functions are based on empirical correlations that utilise information from adjacent cells to estimate flow variables along the wall. These functions help to reduce processing time by bypassing the need to calculate the near-wall region. Bayon-Barrachina and Lopez-Jimenez (2015) [55] highlighted the issues of flow along the wall for accurate findings and suggested to avoid over-refinement of the mesh, as this could compromise the accuracy of the results. Canonsburg (2013) [24] explained the crucial aspect of wall functions involves the non-dimensional distance between the wall and the first node away from the wall (The turbulent boundary layer is composed of three sections: the laminar sub-layer ($y^+ < 5$), the transition layer ($5 < y^+ < 30$), and the turbulence layer ($y^+ > 30$). Ansys Fluent offers a range of options for near wall

treatment in the $k-\varepsilon$ turbulence models, including normal wall function, non-equilibrium wall function, and improved wall treatment. Due to its ability to provide reasonably accurate predictions for most high Reynolds-number (Re), wall-bounded flows and its ability to reduce computational effort by eliminating the need to resolve the near-wall region, the standard wall function was utilised in this study. With the standard wall function, one can easily customise the shear stress and decide whether or not to include roughness on the wall. Throughout the simulations, the wall is considered to be a no-slip wall. It is assumed to have a wall roughness constant of 0.5 and a flow velocity of zero at the wall. The y^+ values for all the grid systems range from 46 to 113, indicating the necessity of using the wall function technique. The inlet Reynold number ranged from 31,654 to 52,861.

3.7 Discretization scheme

When it comes to numerical simulation, discretization of data has a significant influence on the parameters that ultimately shape the results. Based on a study by Bayon-Barrachina and Lopez-Jimenez (2015) [55], it was found that upwind schemes in the RANS model exhibit decreased instability. Therefore, the present investigation opted for the use of upwind schemes. In numerical simulations, the Courant number (Cr) is a dimensionless measurement that indicates the relationship between the time step and the spacing between the computing grid points. It is important to assess the stability of the simulation, as a high Courant number can lead to numerical instability. In the present simulation of the hydraulic jump, the highest Cr used was 0.40, which falls within the safe range for this type of simulation. A widely used algorithm called SIMPLE (Semi-Implicit Method for Pressure-Linked Equations) was used to resolve fluid flow issues. The density and momentum equations were discretised using the first and second-order upwind techniques, respectively.

3.8 Stability & convergence criteria

In this study, the residuals were set to 10^{-6} to analyse the convergence criteria. Due to the abrupt nature of the hydraulic jump, it can be quite challenging to ensure the stability of the numerical model. In this study, a method of trial and error was employed to stabilise the jump by analysing the discharge equilibrium curve for the inflow and

outflow. It becomes evident that the jump's stable conditions were apparent after 11 seconds of time steps.

3.9 Validation of numerical model

Numerical simulation validation is the process of confirming that computer simulation findings accurately reflect the processes being examined in reality. To determine the simulation's accuracy and dependability, experimental data are compared to the results of the simulation. Quantifying the disparities between simulation results and experimental data and locating potential areas for simulation improvement can help to determine the simulation's accuracy and reliability. For validating any numerical model with physical model it must be insure that the numerical model is more or less similar. (Chanson and Gualtieri, 2008) [37] studied the scale effect while analysing the hydraulic jump in two different channel dimensions, they found that some scale, effect the void fraction and bubble count rate, to overcome such issues the present numerical study considered the dimensions of flume and roughness elements identical to physical setup.

3.9.1 Longitudinal velocity variation

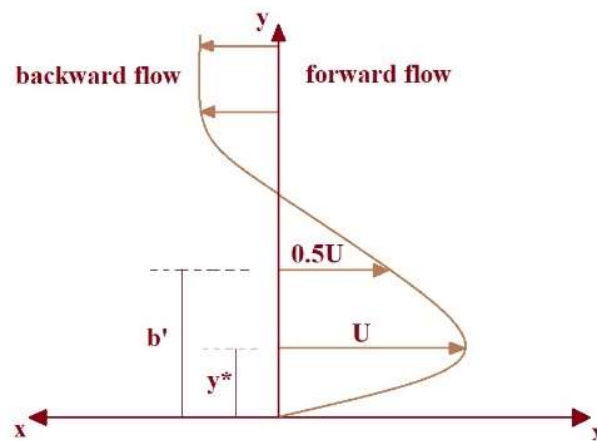


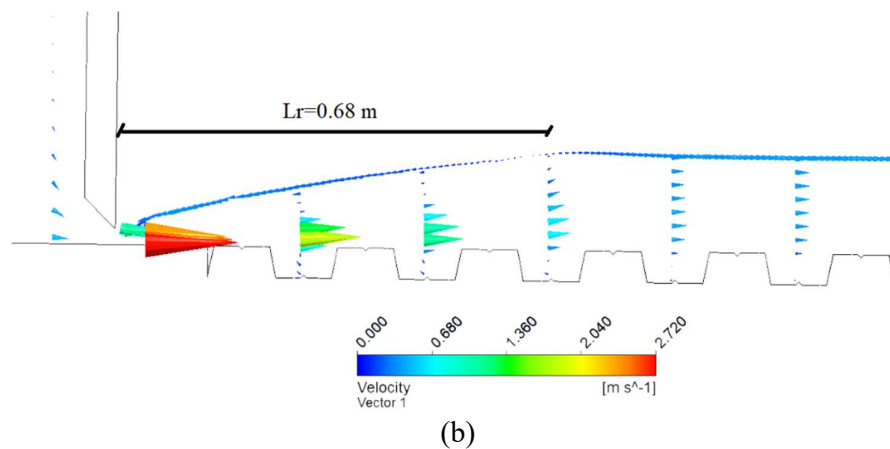
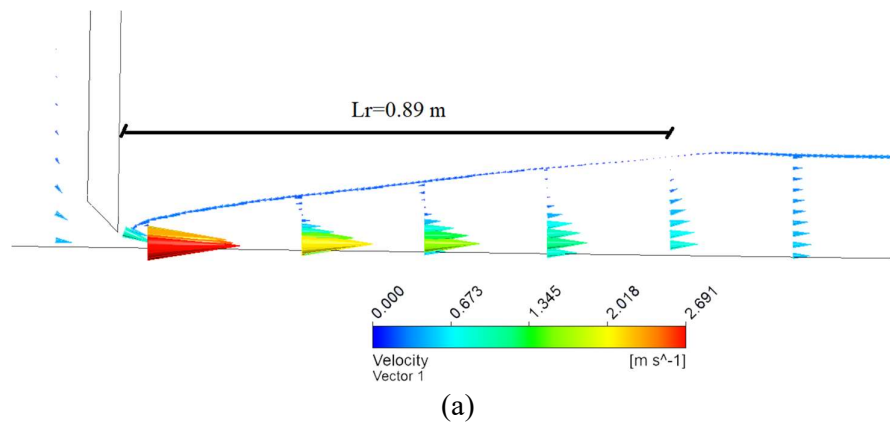
Figure 3.8 Typical longitudinal velocity profile for a hydraulic jump

Figure 3.8 illustrate the general longitudinal velocity profile of developed hydraulic jump in a smooth rectangular channel as discussed by Rajaratnam (1967) [12]. An

inverse relationship between velocity magnitude and the upstream Froude number (Fr_1) was reported by Wang and Chanson (2016) [58].

Rajaratnam (1967) [12] discussed that the maximum velocity observed at a vertical distance of y^* from the bed level, and at a vertical distance of b' , velocity drops to approximately half of the maximum velocity. This is a typical velocity fluctuation in a fully developed roller zone; just above point b' , reverse flow happens due to velocity stagnation.

The standard pictorial representation of the numerically obtained free & submerged hydraulic jump's longitudinal velocity profile on smooth and macrorough beds are shown in Figure 3.9 & Figure 3.10 respectively.



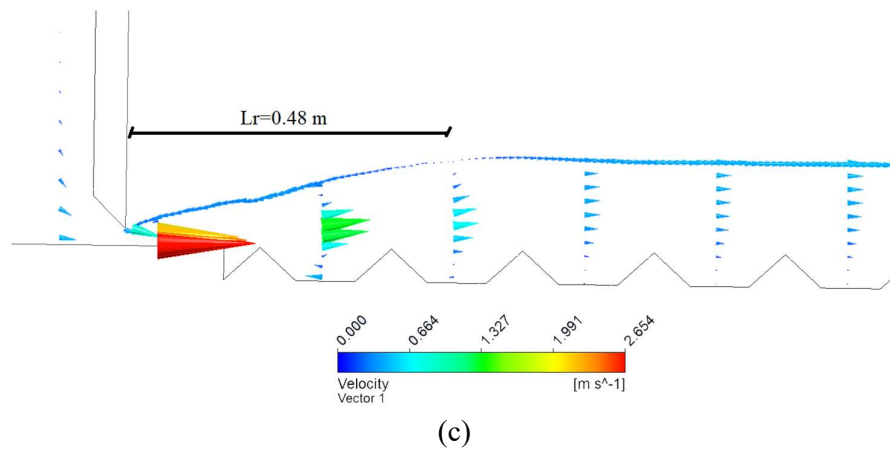
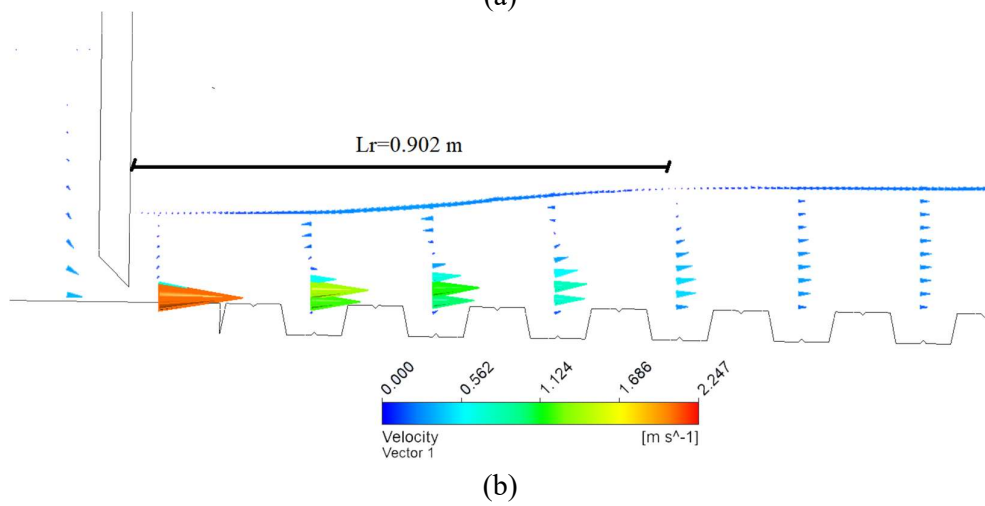
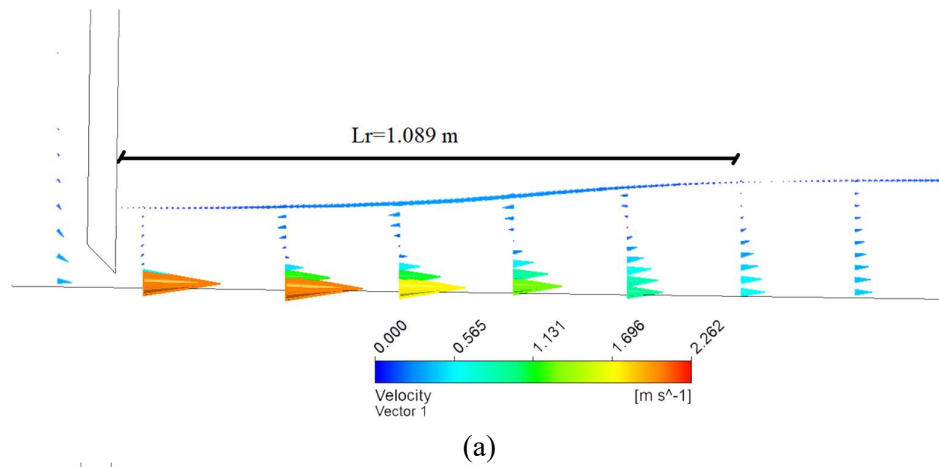


Figure 3.9 Pictorial representation of velocity profile in free hydraulic jump (a) smooth bed (b) trapezoidal macroroughness (c) triangular macroroughness ($Fr_1=5.45$)



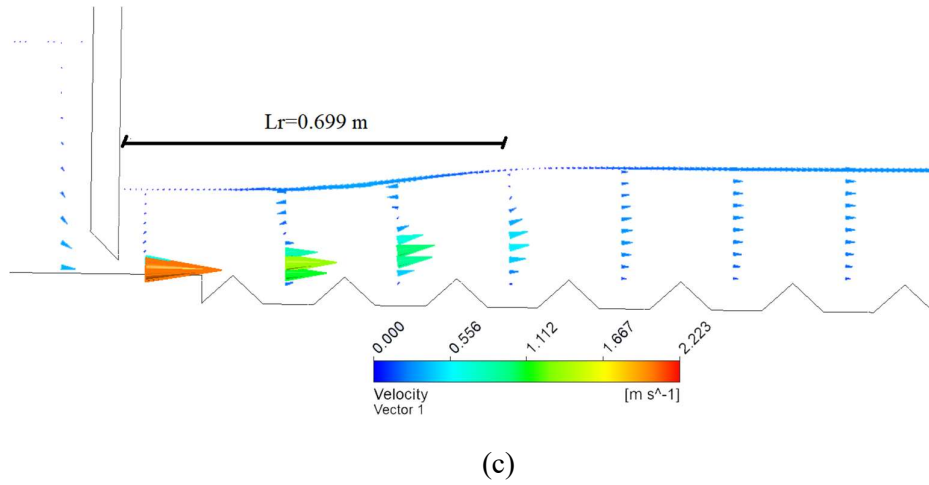


Figure 3.10 Pictorial representation of velocity profile in submerged hydraulic jump
 (a) smooth bed (b) trapezoidal macroroughness (c) triangular macroroughness
 ($Fr_1=6.12$)

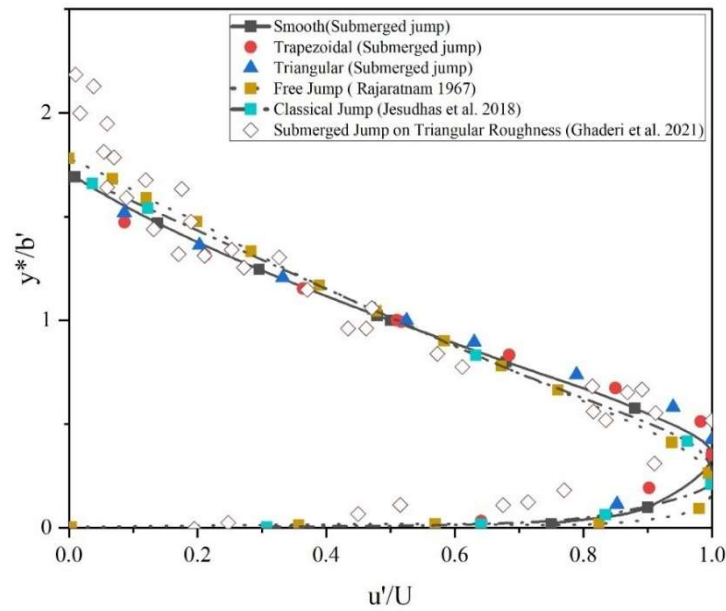


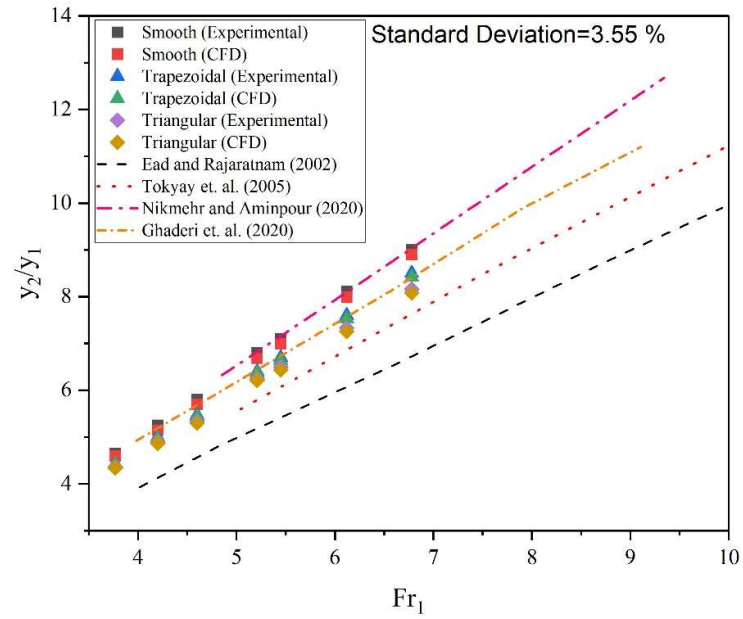
Figure 3.11 Combined dimensionless flow velocity distribution of submerged hydraulic jump for smooth and macroroughness arrangements at $Fr_1=6.1$

By examining the results of Figure 3.9 & Figure 3.10, it can be deduced that the mean flow velocity falls at the topmost region of the gate opening because this is where it is highest. Following the development of the jump, the jump velocity in the roller zone reverses with a stagnation point in a backward direction. Such a description is valid for both macrorough and smooth beds. Velocity distribution obtained for present study well confirms the past velocity distribution of Rajaratnam (1967) [12], Ead and Rajaratnam (2002) [34] and Ghaderi et al. (2021) [18].

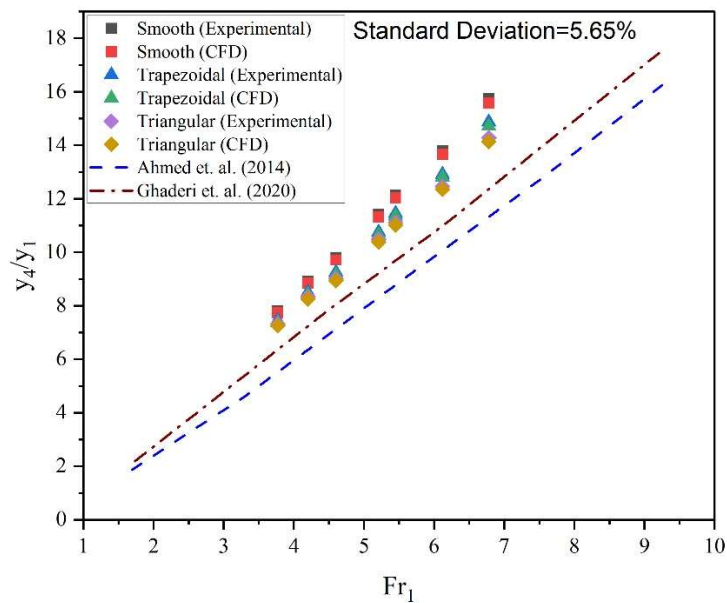
Figure 3.11 represents the dimensionless velocity distribution of submerged hydraulic jump on smooth and macroroughness arrangements for $Fr_1=6.12$. Here, the vertical axis represents the ratio of y^*/b' , where y^* is the distance from the bed and b' is the length scale at which velocity is half of maximum velocity. The horizontal axis represents the ratio of u/U , where u is the time-averaged longitudinal velocity at any section and U is the maximum longitudinal velocity at that section.

The dimensionless velocity distribution graph is typically plotted for the point where the velocity is only half the maximum velocity, which is at a distance of b' from the jump toe. This is due to the downstream region's generally flat velocity distribution, which does not exhibit much variance. Additional computations are made to normalize the velocity values depending on the mean velocity and the height of the jump in order to provide the dimensionless velocity distribution. For validation purposes, previous studies of Ghaderi et al. (2021) [18] are used as the references and presented in comparison with the present findings in Figure 3.11. It is discovered that the trend of dimensionless velocity profiles is more or less similar to the previous research and that the maximum horizontal forward velocity occurred at a shallower flow depth than the depth obtained by Rajaratnam (1967) [12], and Jesudhas et al. (2018) [69].

3.9.2 Dimensionless parameters



(a)



(b)

Figure 3.12 Comparison of a numerical model with experimental model for basic parameters of hydraulic jump (a) y_2/y_1 (b) y_4/y_1

In Figure 3.12, which compares numerical and experimental findings on fundamental jump parameters including the sequent depth ratio (y_2/y_1) and tail-water depth ratio (y_4/y_1), there is good agreement between the two models, with a standard deviation of less than 6%. This degree of agreement demonstrates that CFD modelling may faithfully simulate complex fluid dynamics issues like hydraulic jumps.

$$y_2/y_1 = 1.4569 F_{r1} - 0.8975 \quad R^2 = 0.99 \text{ (smooth bed)} \quad (28)$$

$$y_2/y_1 = 1.3042 F_{r1} - 0.547 \quad R^2 = 0.99 \text{ (macrorough bed)} \quad (29)$$

$$y_4/y_1 = 2.6059 F_{r1} - 2.1254 \quad R^2 = 0.99 \text{ (smooth bed)} \quad (30)$$

$$y_4/y_1 = 2.4229 F_{r1} - 1.8121 \quad R^2 = 0.99 \text{ (macrorough bed)} \quad (31)$$

Further, the comparative evaluation of the Present Study (CFD) against multiple established works Ghaderi et al. (2020), Nikmehr & Aminpour (2020), Ead and Rajaratnam (2002), and Tokyay et al. (2005) confirms the robustness of the CFD approach in capturing the fundamental hydraulic behaviour of free hydraulic jumps. Despite minor discrepancies in the magnitude of the sequent depth ratio (y_2/y_1), primarily attributable to differences in experimental setups, modelling assumptions, and turbulence closure strategies, all studies consistently demonstrate an increasing trend of y_2/y_1 with rising Froude numbers. This agreement reinforces the physical accuracy of the CFD model in reflecting supercritical flow transitions. Notably, the CFD predictions generally fall between the experimental results of Ead and Rajaratnam and the higher values reported by Tokyay et al. (2005), suggesting that the numerical model successfully balances between under and over prediction. These insights not only validate the present CFD methodology but also underscore the importance of model calibration and sensitivity to roughness configurations and turbulence treatments for achieving improved predictive fidelity in future investigations.

The comparative analysis of the submerged hydraulic jump characteristics highlights meaningful trends and deviations between the present CFD study and previous experimental investigations. The consistently higher sequent depth ratio (y_4/y_1)

predicted in comparison to Ahmed et al. (2014), with a systematic deviation ranging between 1.7 and 2.8 and a relatively low standard error of 0.140, suggests a moderate yet reliable overestimation by the CFD model. This indicates a potential influence of model assumptions or boundary conditions on the simulated flow structure. Conversely, the comparison with Ghaderi et al. (2020) yields a standard error of 0.111, reflecting a relatively close agreement and reinforcing the predictive capability and reliability of the present CFD framework under varying Froude number regimes. Overall, while minor pointwise differences exist, the CFD model demonstrates a strong potential for accurately simulating submerged hydraulic jumps, particularly when validated against recent experimental data.

CHAPTER 4: RESULT & DISCUSSION

4.1 General

This chapter is concerned with the various boundary conditions which may be chosen in dissimilar flume conditions in order to analyse the hydraulic jump phenomenon numerically and to understand the influence of dissimilar shape of artificial macroroughness on hydraulic jump characteristics.

4.2 Effect of various flume configurations on characteristics of classical hydraulic jump

In past studies researchers used different inlet and outlet boundary conditions to perform numerical simulation. On observing the past numerical studies to investigate the hydraulic jump phenomenon it was noticed that the hydraulic jump phenomenon can be studied numerically by creating different types of flumes with appropriate boundary conditions. Earlier work of Bayon-Barrachina and Lopez-Jimenez (2015) [55] considered two types of flumes to investigate the hydraulic jump phenomenon in a smooth rectangular channel and found negligible effect on jump characteristics however the computational cost was reduced significantly.

This study considered two different approaches to prevail the initial jump conditions. In first approach the effect of upstream water head is considered and in second approach same is neglected. In this regard, four different geometries considering different arrangement in downstream were created for first and second approach separately. Therefore, this section analysed eight feasible flumes.

First approach: In Model I, flume created similar to the laboratory flume where inlet sluice gate and outlet tailgate is equipped. In Model II & III, flume created where inlet sluice gate is provided but in downstream, weir in form of obstruction is provided instead of tailgate. In Model IV, flume is created with inlet sluice gate but without any obstruction in downstream. Figure 4.1 illustrate the different models of flume considered for first approach.

Boundary condition used in first approach:

- (a) Inlet: $u = u_1, \frac{\partial p}{\partial x} = 0, \alpha = f(y), k = k_{inlet}, \varepsilon = \varepsilon_{inlet}$
- (b) Outlet for Model I, II & III: $\frac{\partial p}{\partial x_i} = 0, p = 0, \frac{\partial \alpha}{\partial x_i} = 0, \frac{\partial k}{\partial x_i} = 0, \frac{\partial \varepsilon}{\partial x_i} = 0$
- (c) Outlet for Model IV: $u = f(y), p = f(y), \alpha = f(y), \frac{\partial k}{\partial x_i} = 0, \frac{\partial \varepsilon}{\partial x_i} = 0$
- (d) Atmosphere: $\frac{\partial u}{\partial x_i} = 0, p = 0, \frac{\partial \alpha}{\partial x_i} = 0, \frac{\partial k}{\partial x_i} = 0, \frac{\partial \varepsilon}{\partial x_i} = 0$
- (e) Wall & Floor: $u = 0, \frac{\partial p}{\partial x_i} = 0, \frac{\partial \alpha}{\partial x_i} = 0, k = f(y), \varepsilon = f(y)$

Second approach: In Model V, flume created without inlet sluice gate but equipped with tailgate as an obstruction to form the jump. In Model VI & VII, flume created without inlet sluice gate but equipped with weir as an obstruction instead of tailgate. In Model VIII, flume created without inlet sluice gate and without any obstruction in downstream. Figure 4.2 illustrate the different models of flume considered for second approach.

Boundary condition used in second approach:

- (a) Inlet: $u = f(y), p = f(y), \alpha = f(y), k = k_{inlet}, \varepsilon = \varepsilon_{inlet}$
- (b) Outlet for Model V, VI & VII: $\frac{\partial p}{\partial x_i} = 0, p = 0, \frac{\partial \alpha}{\partial x_i} = 0, \frac{\partial k}{\partial x_i} = 0, \frac{\partial \varepsilon}{\partial x_i} = 0$
- (c) Outlet for Model VIII: $u = f(y), p = f(y), \alpha = f(y), \frac{\partial k}{\partial x_i} = 0, \frac{\partial \varepsilon}{\partial x_i} = 0$
- (d) Atmosphere: $\frac{\partial u}{\partial x_i} = 0, p = 0, \frac{\partial \alpha}{\partial x_i} = 0, \frac{\partial k}{\partial x_i} = 0, \frac{\partial \varepsilon}{\partial x_i} = 0$
- (e) Wall & Floor: $u = 0, \frac{\partial p}{\partial x_i} = 0, \frac{\partial \alpha}{\partial x_i} = 0, k = f(y), \varepsilon = f(y)$

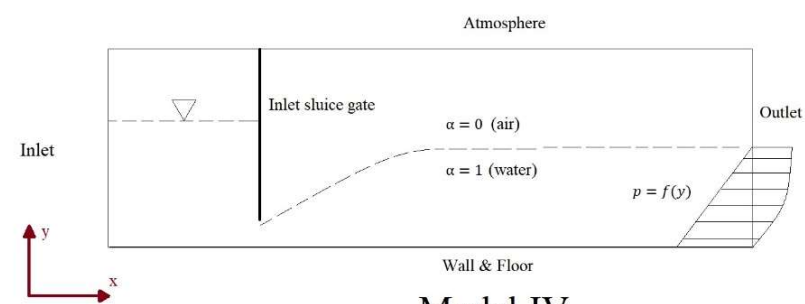
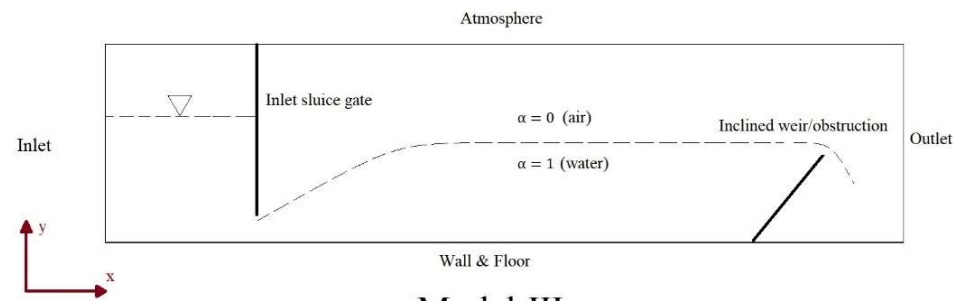
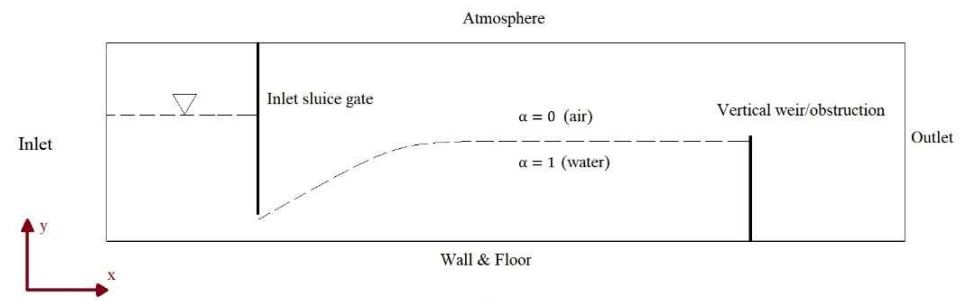
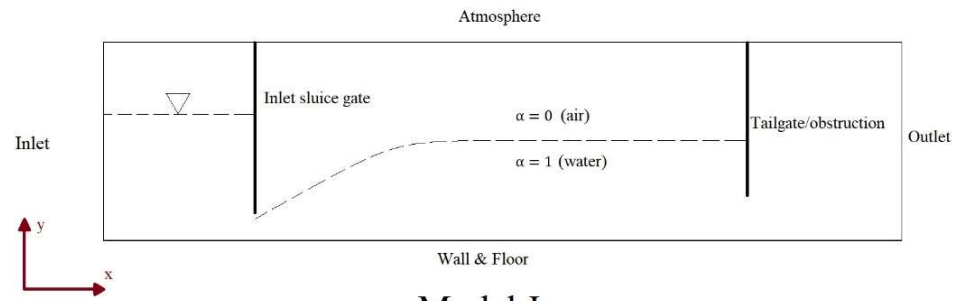


Figure 4.1 Sketch of different types of flumes used in first approach

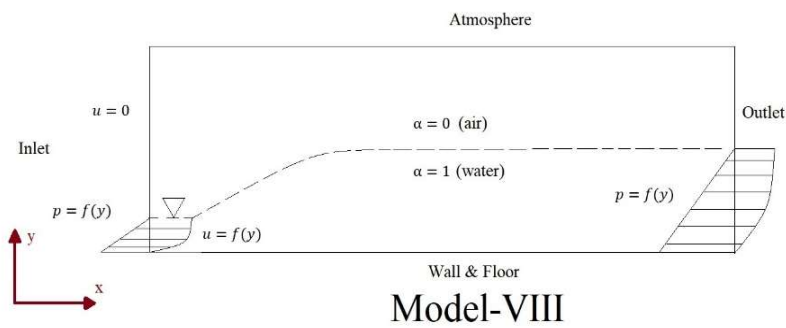
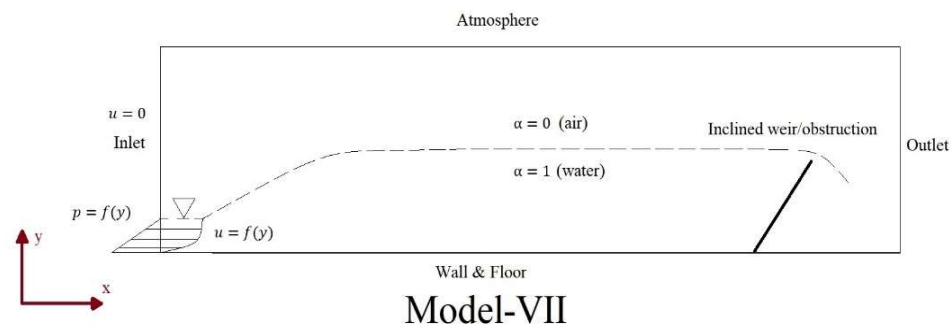
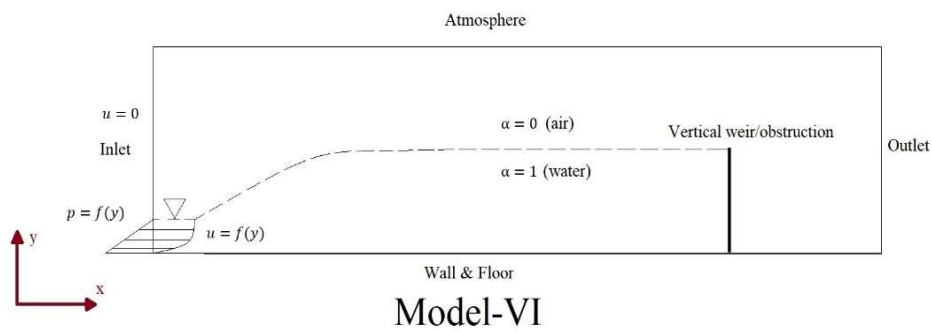
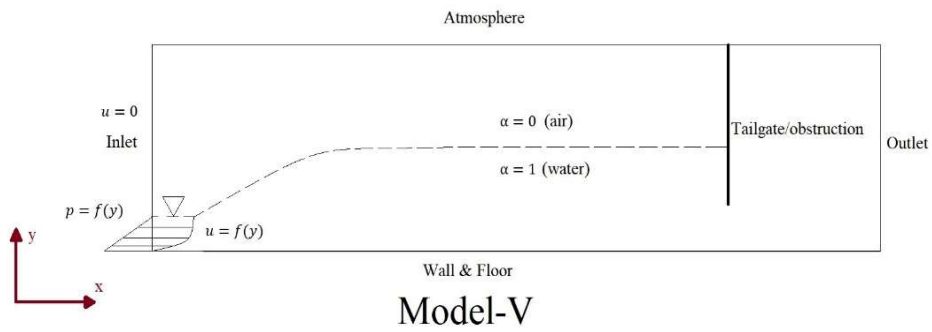
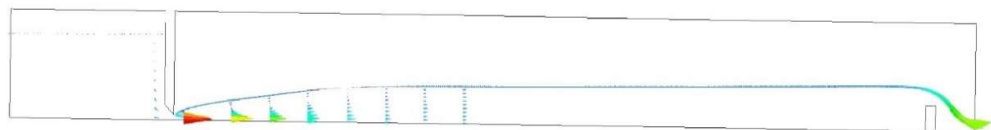


Figure 4.2 Sketch of different types of flumes used in second approach

4.2.1 CFD generated flow profile including longitudinal velocity profile of hydraulic jump



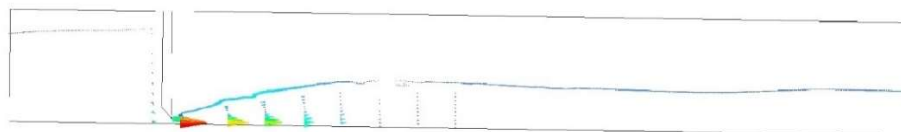
Model-I
(Computational time=16 hrs.)



Model-II
(Computational time=13 hrs.)



Model-III
(Computational time=13 hrs.)

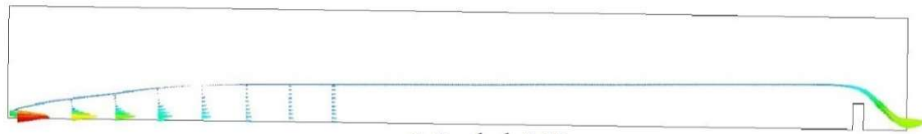


Model-IV
(Computational time=09 hrs.)

Figure 4.3 Numerically simulated flow profile & longitudinal velocity profile at different sections of flume for first approach



Model-V
(Computational time=13 hrs.)



Model-VI
(Computational time=10.5 hrs.)



Model-VII
(Computational time=10.5 hrs.)



Model-VIII
(Computational time=07 hrs.)

Figure 4.4 Numerically simulated flow profile & longitudinal velocity profile at different sections of flume for second approach

Figure 4.3 & Figure 4.4 illustrate the flow profiles & longitudinal velocity profiles at different section within hydraulic jump for each model and represents the examples of hydraulic jump simulated using possible approaches. It can be clearly identified that the flow profiles are almost similar in nature and no major variations are reported. Hence, when comparing them closely between each other there remains no substantial impact on the accuracy of the model outcomes. There are no undesirable effects including wave formation despite the fact that the new approach means moving the boundary conditions much closer to the phenomenon of interest. The velocity profiles are almost similar for most of the models but small variation is seen in last model of the both cases. However, these small variation in model disappears on averaging the velocities. However, the computational time in all four models is not same which may be a major outcome of providing different boundary conditions to the outlet. Domain reduction helps to reach computational times up to 50% shorter in some cases which also correlates to the statements made by Bayon-Barrachina and Lopez-Jimenez (2015) [55].

4.3 Comparative analysis of eight (08) different types of strip macroroughness configurations

The incorporation of various artificial strip macroroughness shapes such as triangular, rectangular, trapezoidal, and semicircular within stilling basins is a deliberate hydraulic engineering strategy that may enhance energy dissipation, stabilize hydraulic jumps, and help control turbulence. Each shape may uniquely disturb the flow field, potentially influencing roller length, sequent depth, jump position, and air entrainment characteristics. For instance, triangular and trapezoidal shapes could promote rapid jump initiation and upward flow deflection, thereby increasing turbulence and energy loss. Rectangular elements may provide strong obstruction and pronounced wake formation, while semicircular shapes might encourage smoother recirculation and reduce local flow separation making them potentially effective in minimizing sediment deposition and local scour.

The selection of roughness shape may also depend on practical factors such as ease of construction, cavitation resistance, and long-term durability. Rectangular and semicircular shapes are often preferred due to their fabrication simplicity and

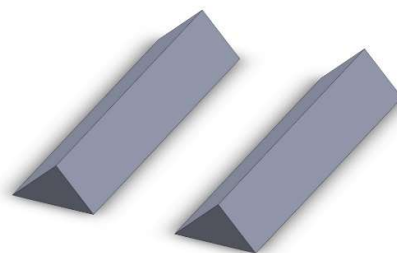
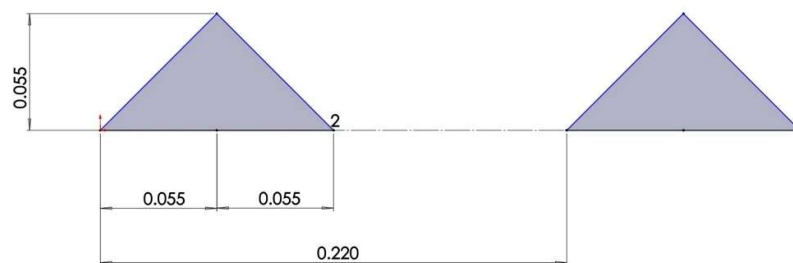
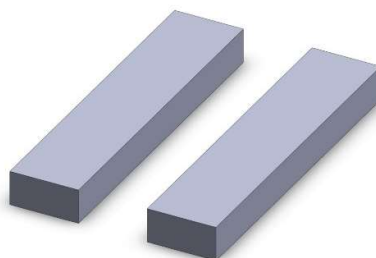
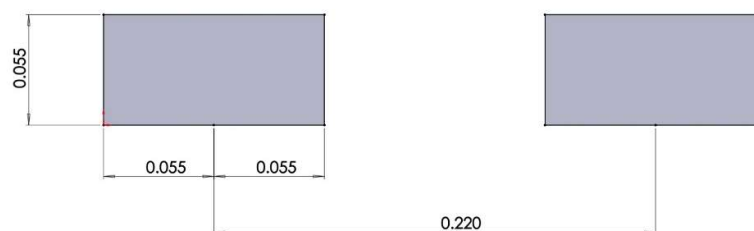
maintenance advantages. Ultimately, the choice of macroroughness geometry should be tailored to site-specific flow conditions and design objectives, aiming to ensure maximum hydraulic efficiency, structural stability, and operational sustainability.

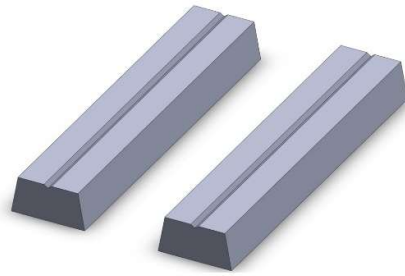
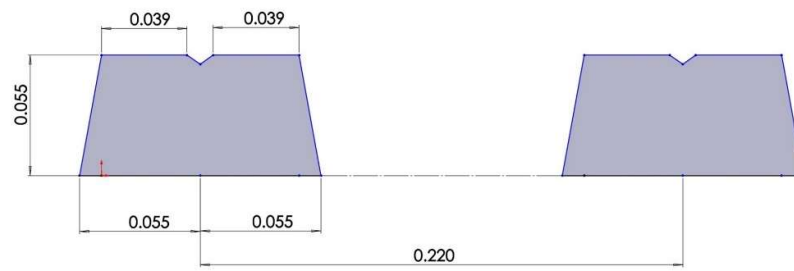
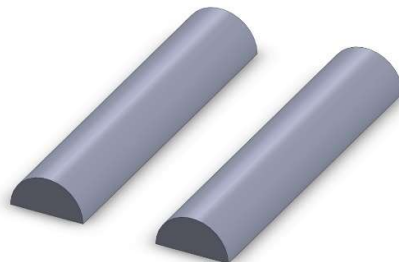
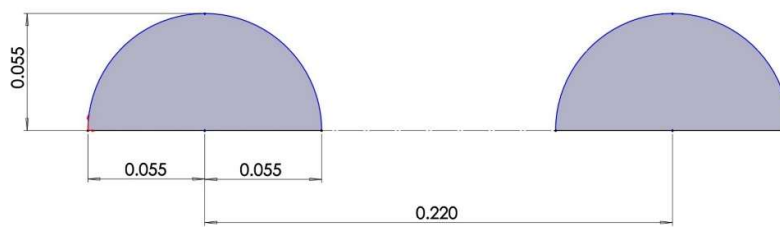
During this investigation, macrorough strips are positioned across the flow so that they would be most effective. This design is selected according to accepted hydraulic concepts, literature on the topic and stability criteria. Having perpendicular strips strategically placed inside the stilling basin helps break the water's momentum and makes it dissipate better.

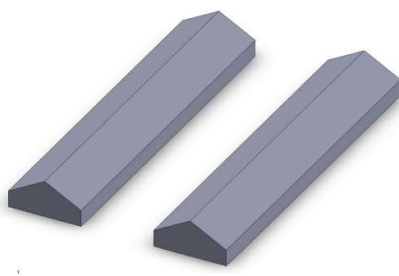
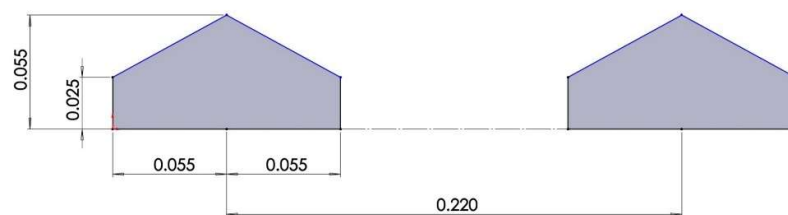
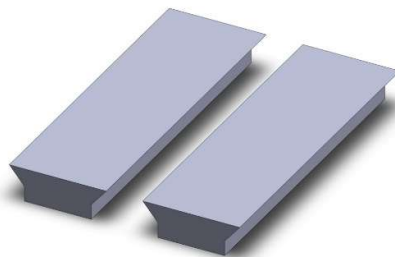
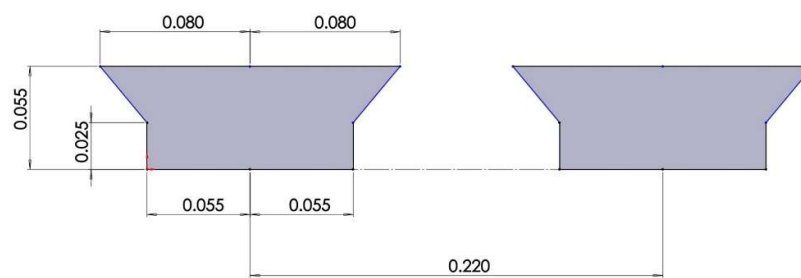
Additionally, this arrangement supports the formation of a strong roller and increased air entrainment, improving oxygen transfer and aeration efficiency. It also aligns with practical and field-tested designs such as U.S. Bureau of Reclamation Type II basins use flow-normal energy dissipating features such as baffle blocks, sills, or roughness strips in this manner. The perpendicular orientation therefore provides comparative relevance and design continuity with field-tested systems.

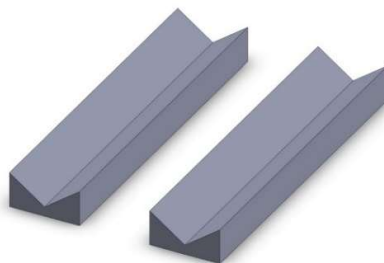
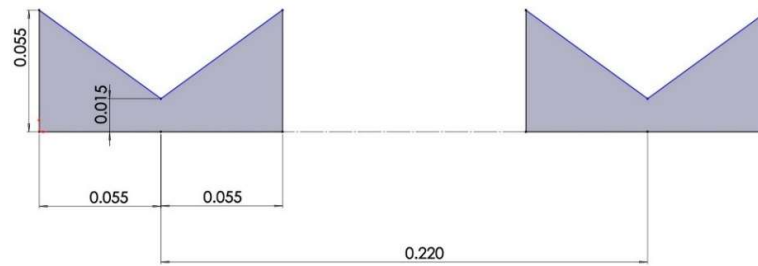
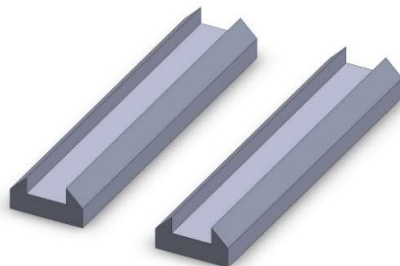
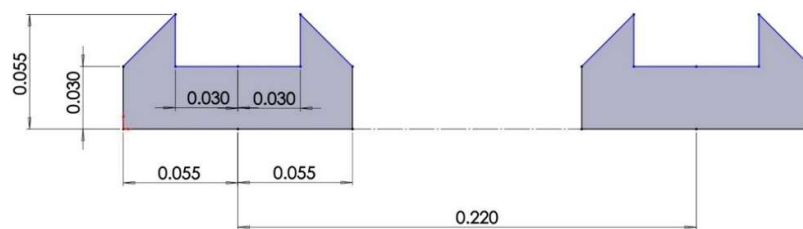
While alternative orientations such as angled, parallel, or staggered arrangements may influence flow behaviour differently potentially causing asymmetric flow separation, uneven roller development, or reduced turbulence generation.

Figure 4.5 illustrate the eight (08) different types of strip macroroughness models *i.e.*, S₁, S₂, S₃, S₄, S₅, S₆, S₇, S₈ for the analysis of the free and submerged hydraulic jump characteristics. Here, eight different possible structures in form of strips are chosen and spread to the downstream bed of the hydraulic structure. In previous studies strip macroroughness was found beneficial to improve the jump characteristics. However, past studied investigated only few possible strip macroroughness shapes.

(a) Model S₁(b) Model S₂

(c) Model S₃(d) Model S₄

(e) Model S₅(f) Model S₆

(g) Model S₆(h) Model S₈

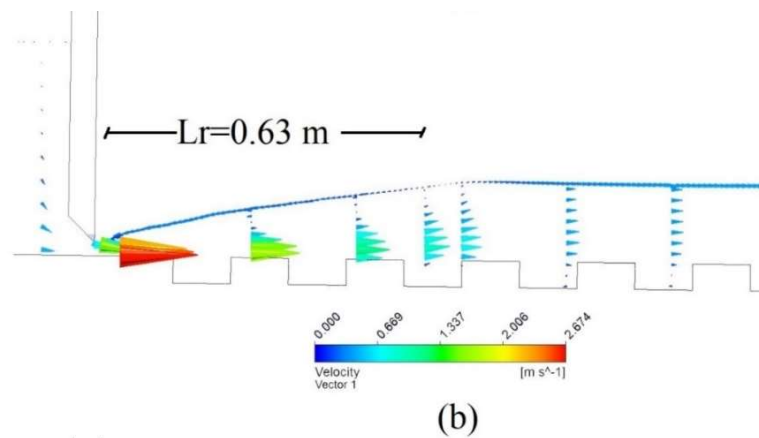
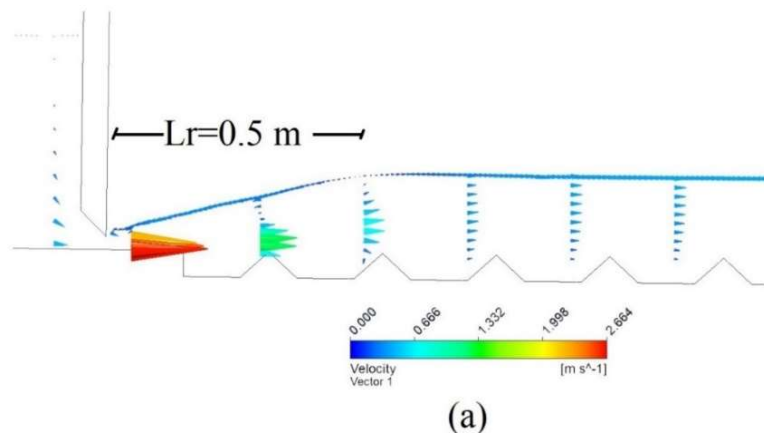
(Note: Above dimensions are in meter)

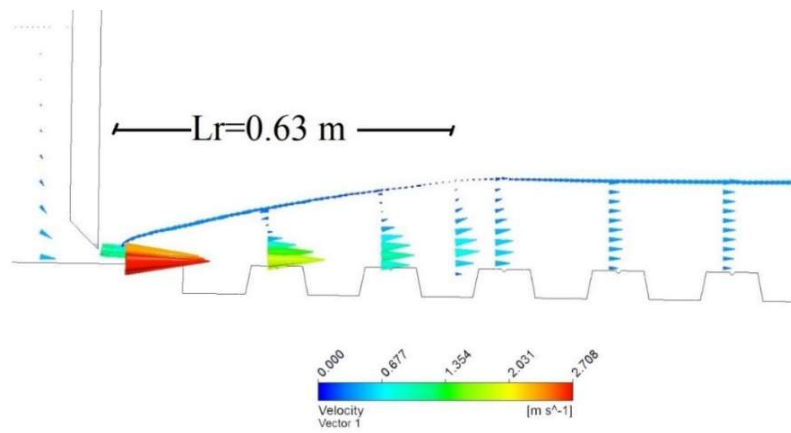
Figure 4.5 Model sketch of adopted different strip macroroughness shapes

4.3.1 Flow profiles & Velocity profile

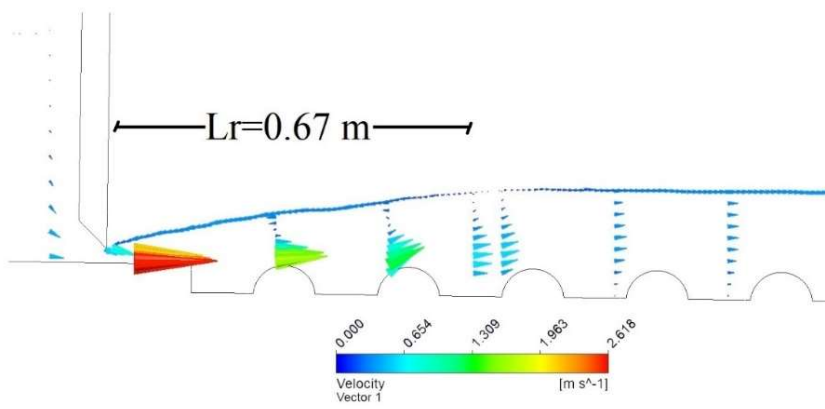
This section shows the flow profiles and longitudinal velocity profiles of free and submerged hydraulic jump over various strip macroroughness *i.e.*, S_1 , S_2 , S_3 , S_4 , S_5 , S_6 , S_7 , S_8 downstream of the hydraulic structure. This section also include the values of length of roller for each case.

4.3.1.1 Flow profile and velocity variation in case of free hydraulic jump

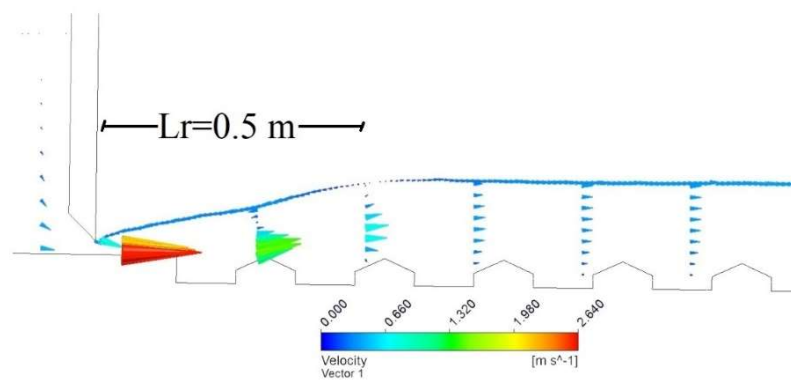




(c)



(d)



(e)

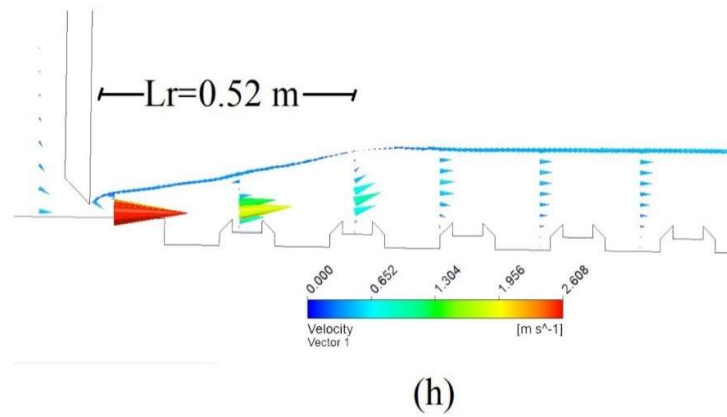
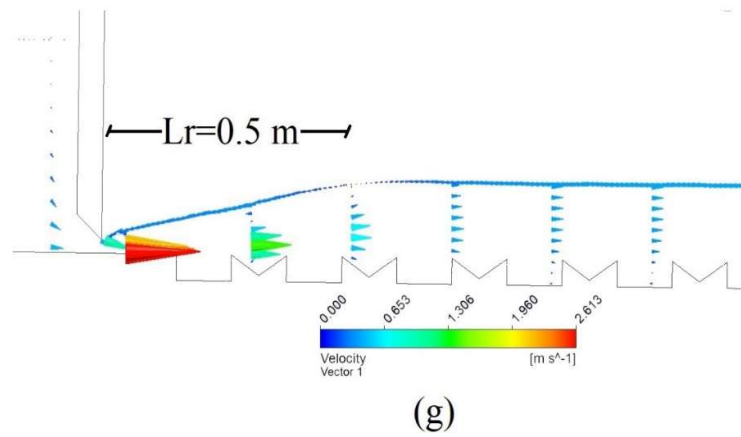
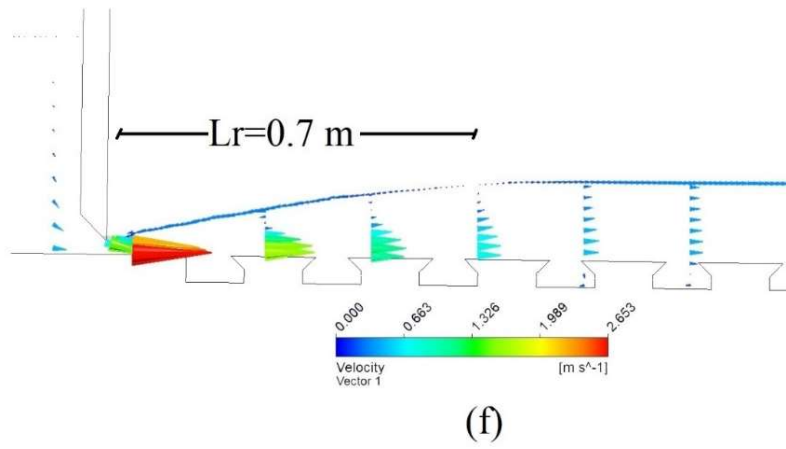
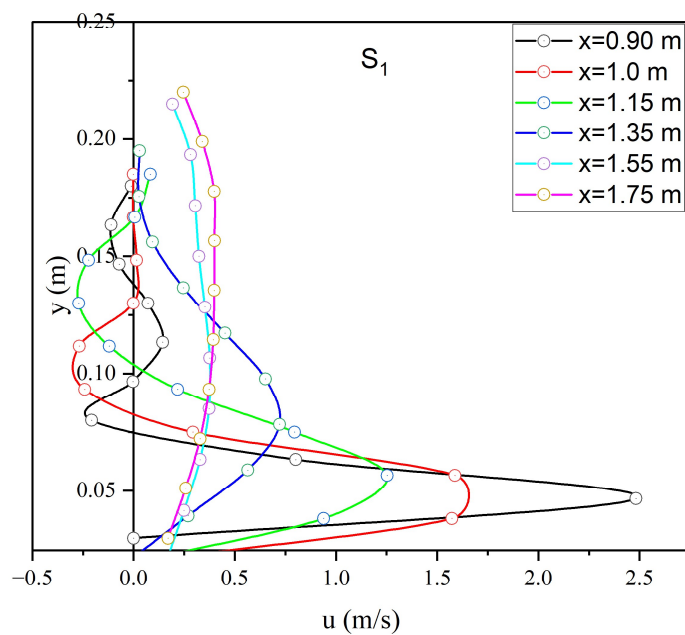
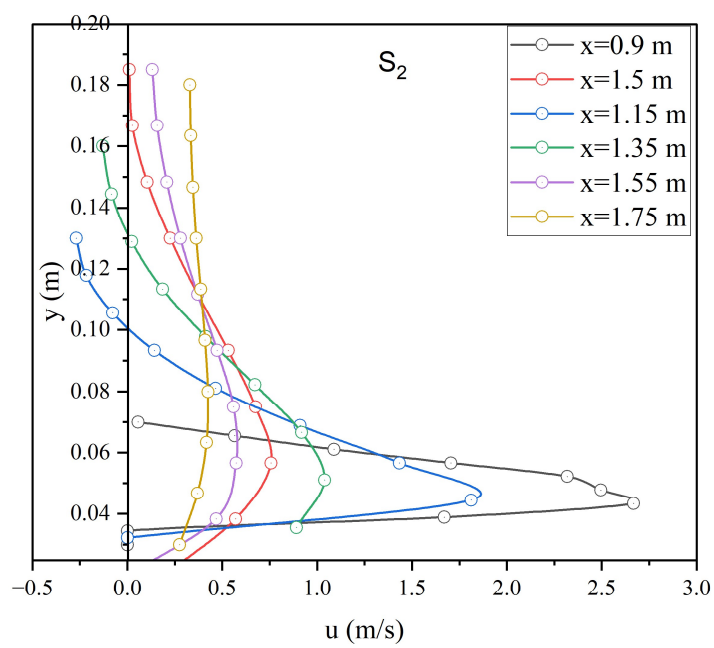


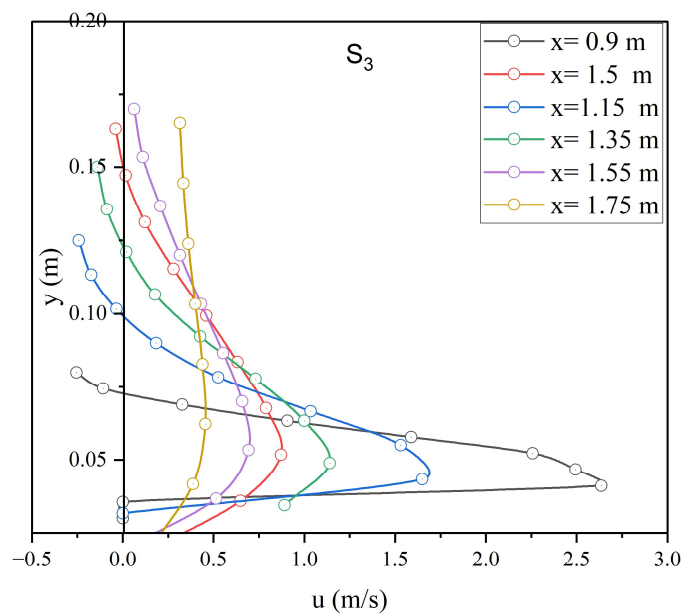
Figure 4.6 (a-h) Numerically generated flow profiles showing longitudinal velocities at different sections for each model for free hydraulic jump.



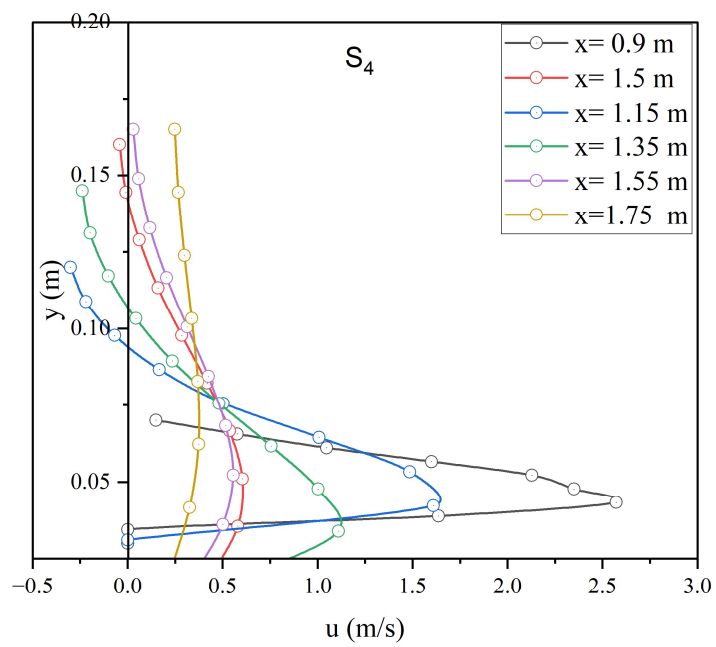
(a)



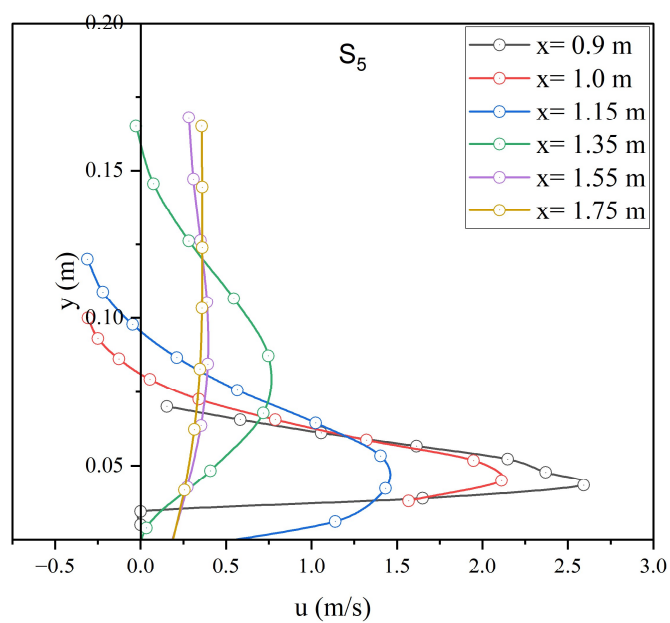
(b)



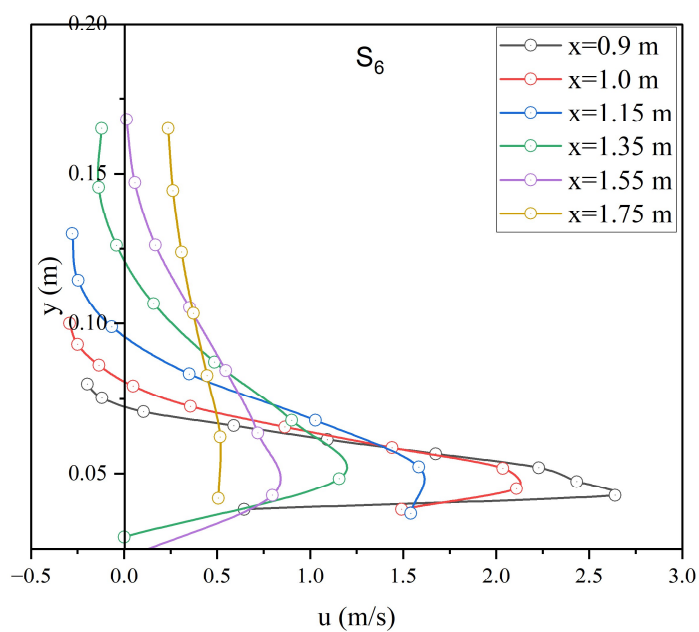
(c)



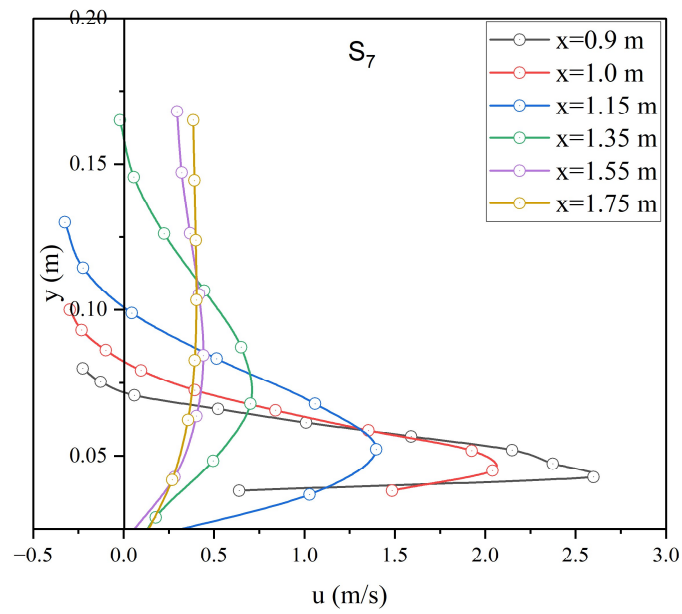
(d)



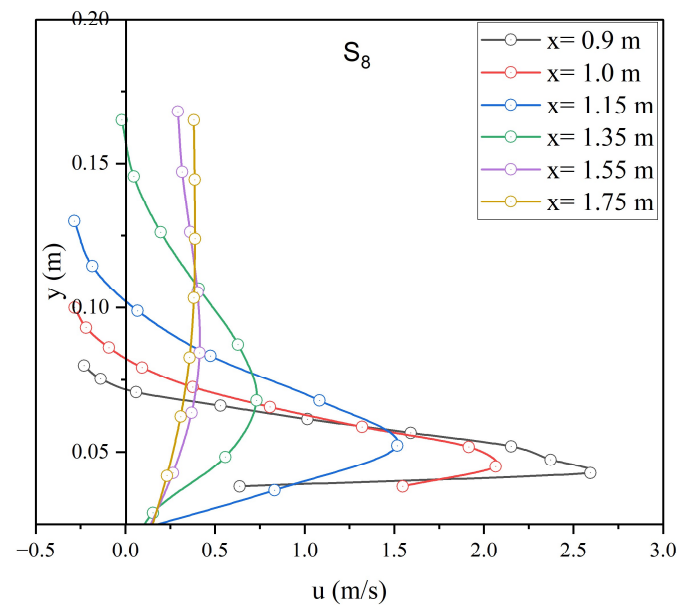
(e)



(f)



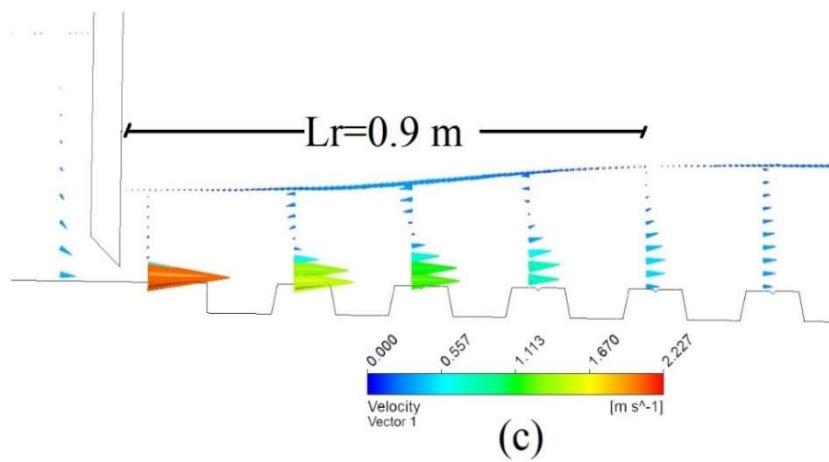
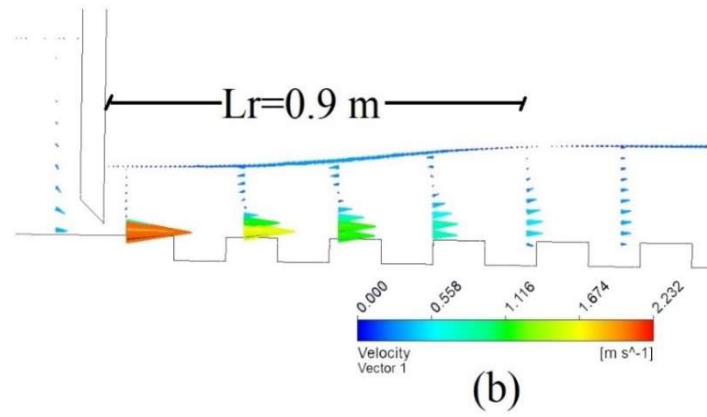
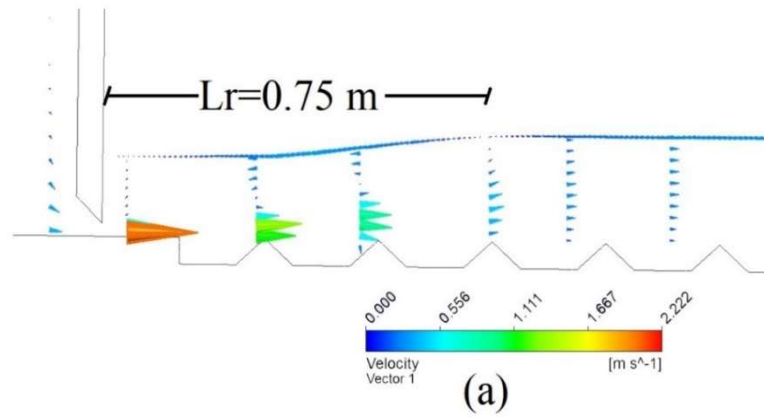
(g)

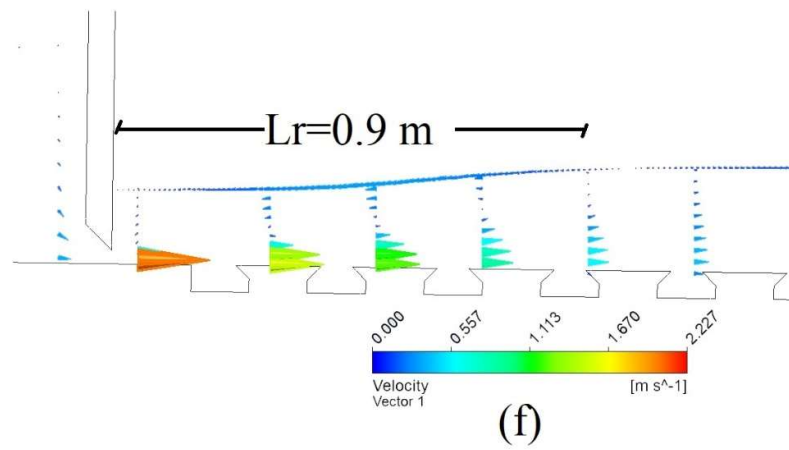
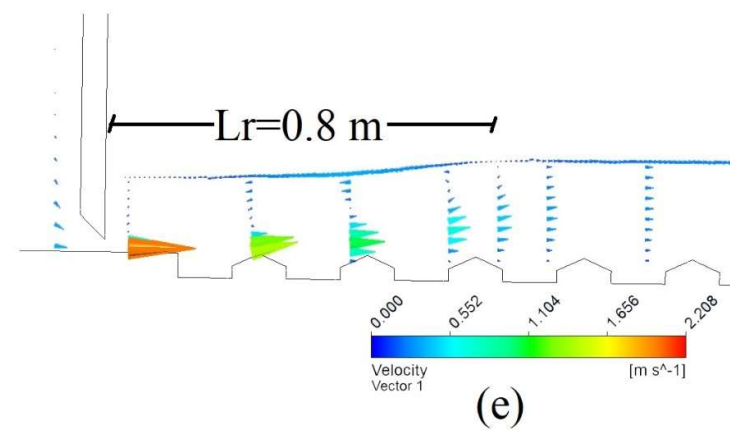
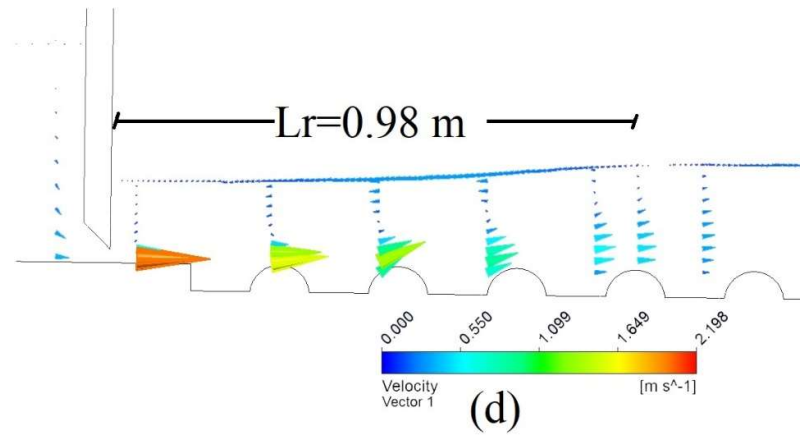


(h)

Figure 4.7 (a-h) Graphical representation of longitudinal velocity profiles of free hydraulic jump

4.3.1.2 Flow profile and velocity variation in case of submerged hydraulic jump





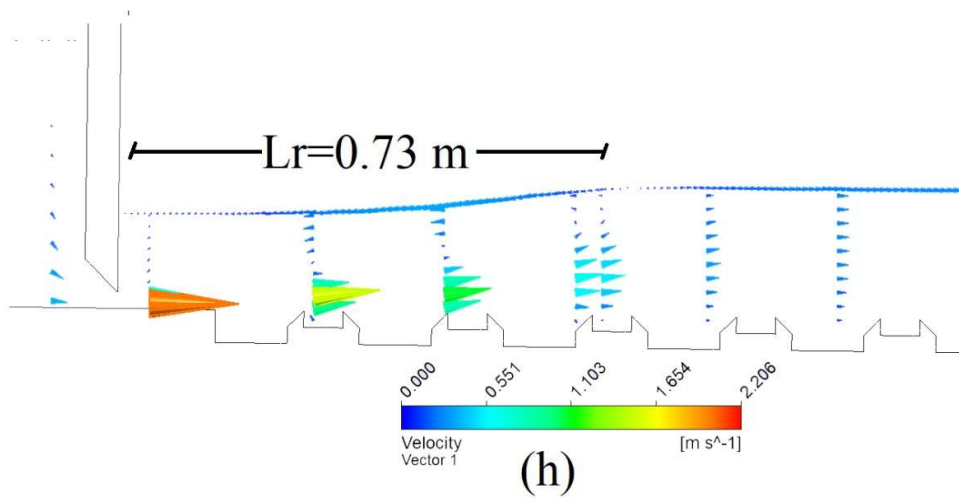
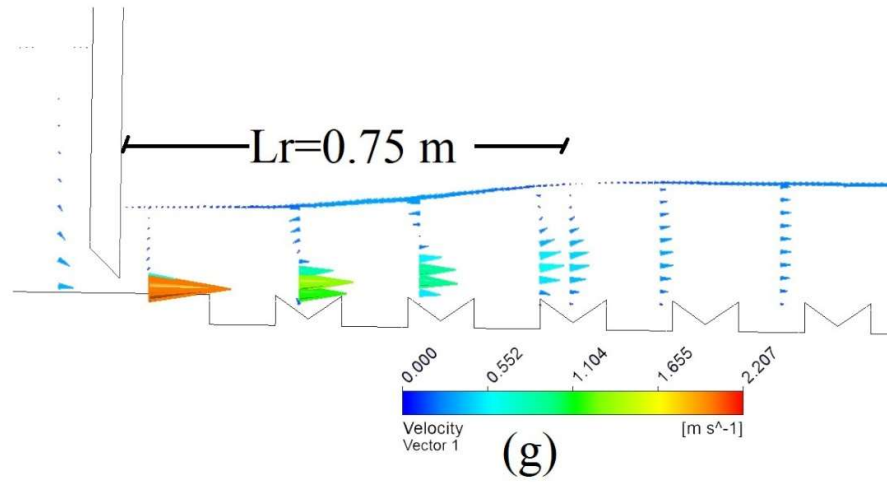
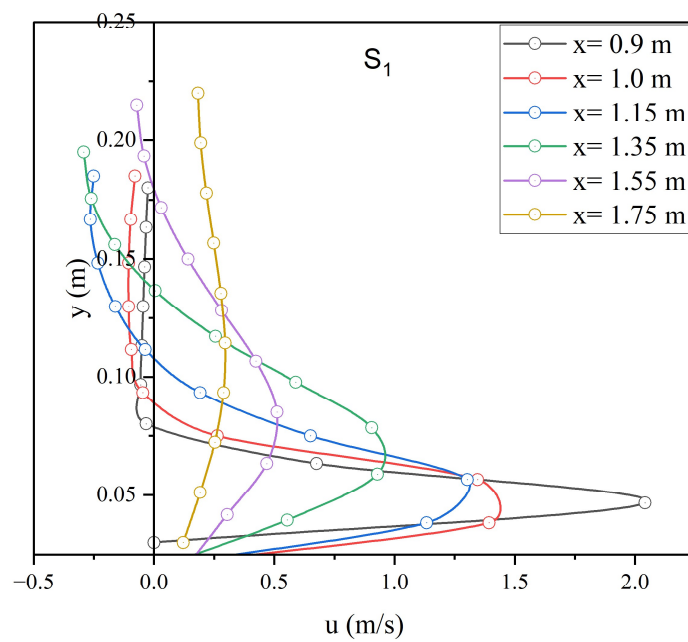
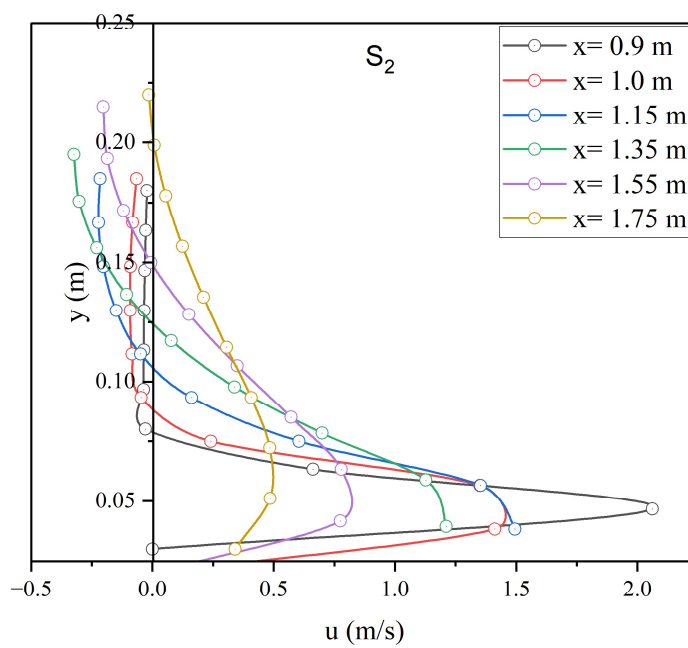


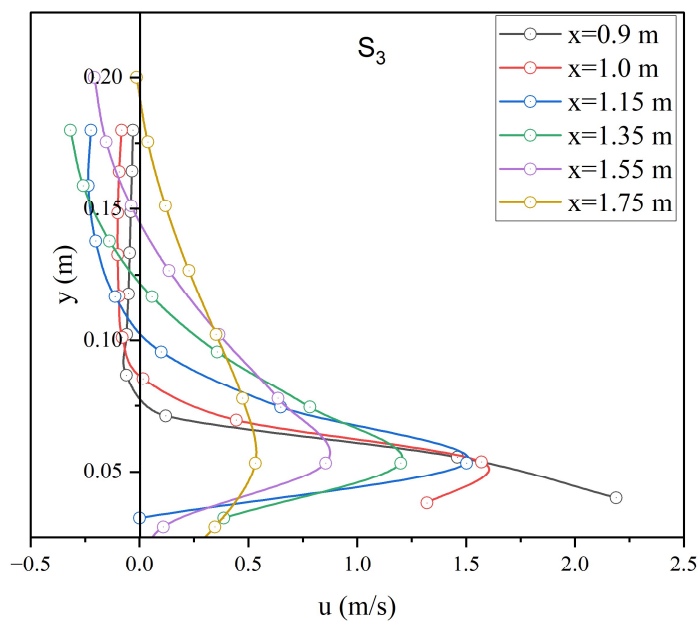
Figure 4.8 (a-h) Numerically generated flow profiles showing longitudinal velocities at different sections for each model for submerged hydraulic jump



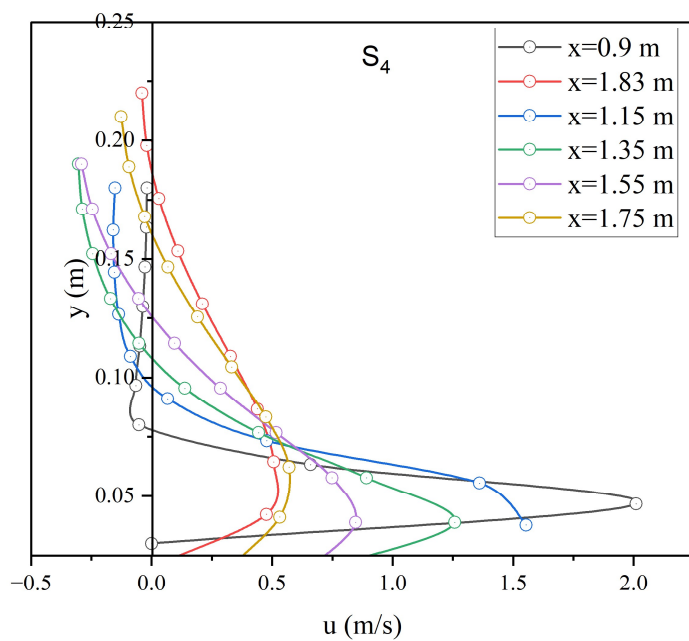
(a)



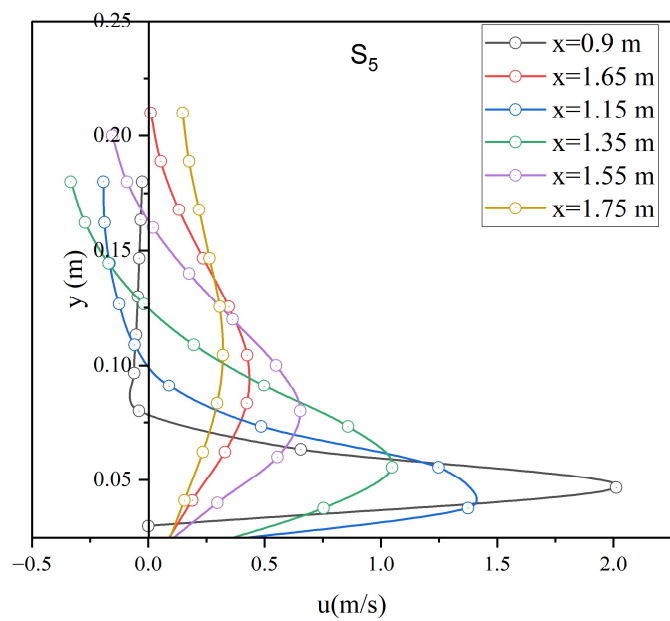
(b)



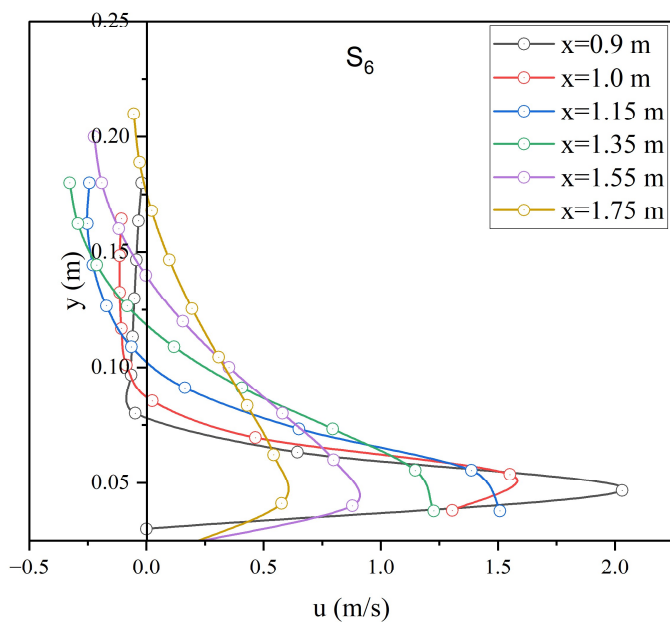
(c)



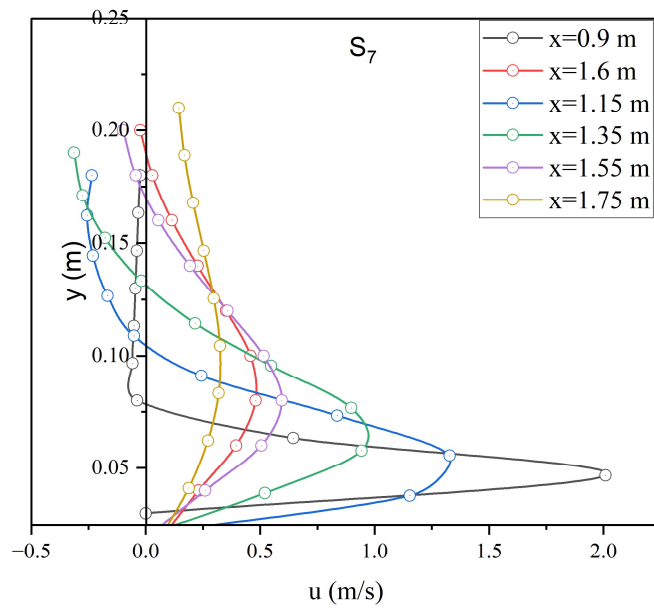
(d)



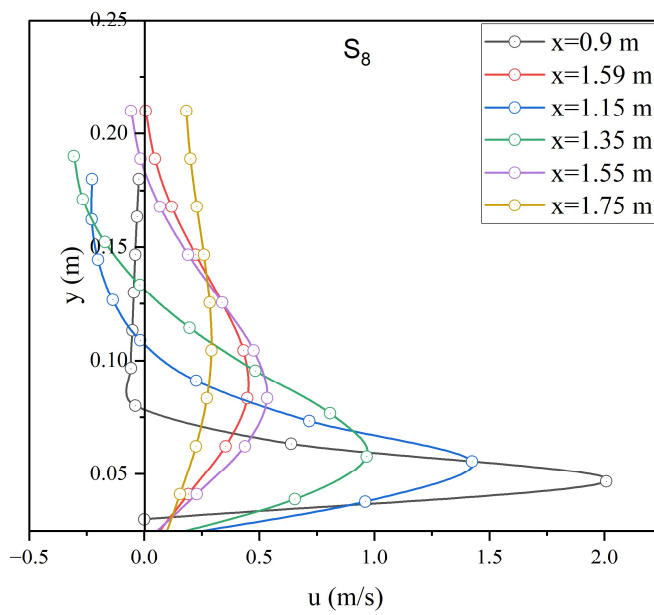
(e)



(f)



(g)



(h)

Figure 4.9 Graphical representation of longitudinal velocity profiles of submerged hydraulic jump

Figure 4.6 & Figure 4.8 illustrate the flow profiles and velocity distribution at different sections of flume. From the results of Figure 4.6 & Figure 4.8, as expected, the flow profiles and velocity distributions are almost similar in nature and in good agreement with the previous findings of Ghaderi et al. (2021) [18].

Due the contracted jet below the gate and flow, longitudinal velocities are less at the upper part of the gate opening, because the magnitude of longitudinal flow velocity is more at the vicinity of gate. A rise in reverse flow phenomena is observed in the vicinity of gates over the time and then begins to decline and represents the stagnation zone on the surface of the atmosphere. In the case of free and submerged hydraulic jump the observations were also used to infer that the recirculation area takes place near the sluice gate. Further it is observed that the negative magnitude of velocity is higher in mid of the recirculation zone than near the sluice gate. After propagations of fluid the magnitude of the velocity becomes positive and somehow higher positive magnitudes are observed at free levelled surface. The above statements are true for both smooth and macrorough beds. In macrorough beds it is identified that the velocity magnitude in vicinity of two consecutive macroroughness regions are quite less than average velocity.

Figure 4.7 & Figure 4.9 shows the graphical representation of longitudinal velocity distribution at six different sections in case of free and submerged hydraulic jump respectively. These sections are the same for each arrangement of roughness. The velocity was measured from the bottom of the bed and up to the free surface. The velocity profile follows the nature of past studies of hydraulic jump in which the velocity varies parabolically up to certain length in the direction of flow due to upcoming water jet and the due to circulation and huge turbulence the velocity decay starts and the graph follow a backward movement. Further a moment comes when the velocity becomes almost zero for a while and becomes negative in the roller regions.

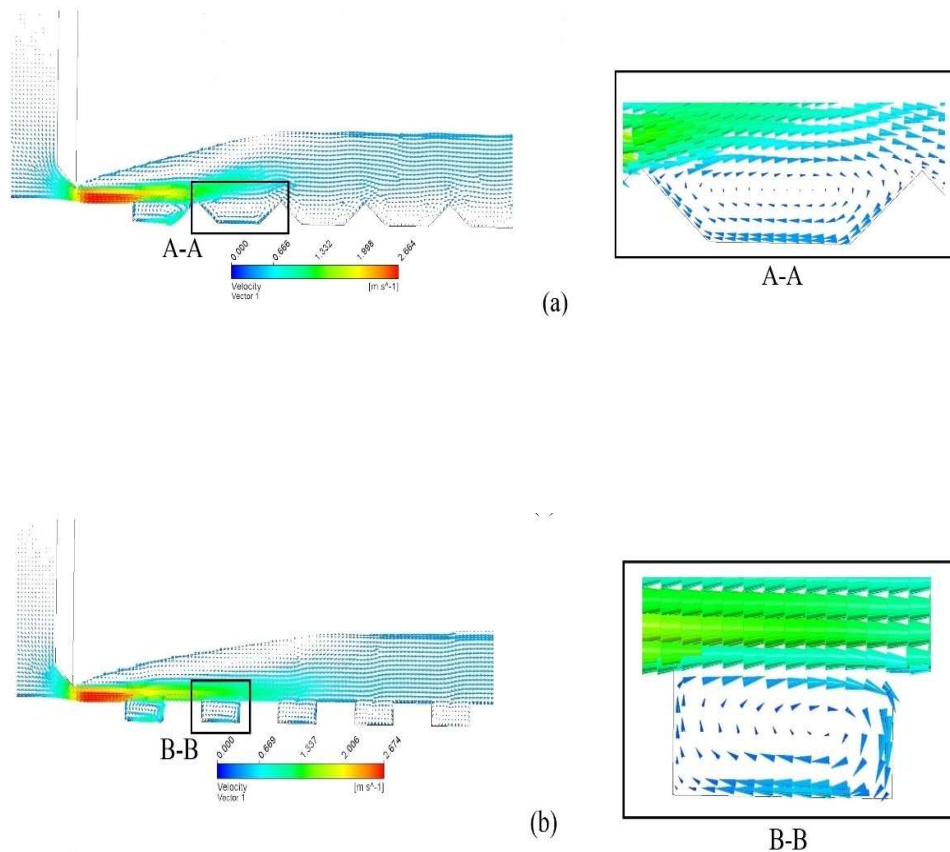
4.3.2 Velocity vector variation

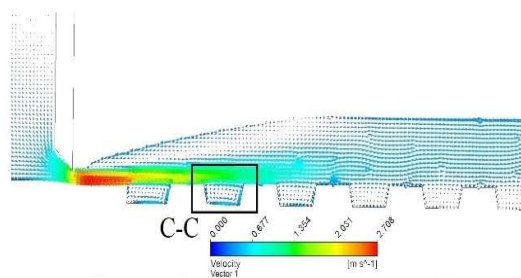
A velocity vector is a mathematical representation used in physics to describe the motion of an object or fluid particle. It is a vector quantity because it has both magnitude and direction. In fluid dynamics, velocity vectors are often used to depict

the flow of a fluid at various points in space. The velocity vector is normally indicated in diagrams by a line with an arrow-head, where the length of the line gives a measure of the velocity and the direction of the arrow head gives a measure of the direction of motion.

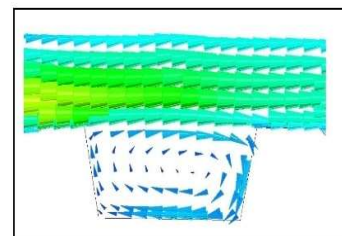
In fluid dynamics, the concepts of velocity vectors play a significant role in determining the representation of flow fields in the form of vectors, assessment of the behaviour of fluids in given conditions. Velocity vectors are generally acquired from measurements through flow visualization or computational modelling. Below section represented the velocity vector to know the behaviour of individual fluid particle over macrorough beds in case of hydraulic jump.

4.3.2.1 Free hydraulic jump

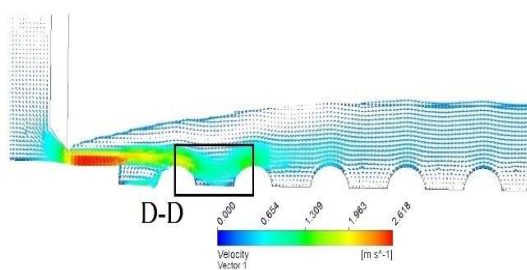




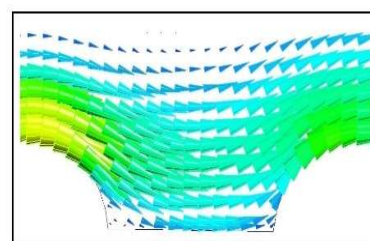
(c)



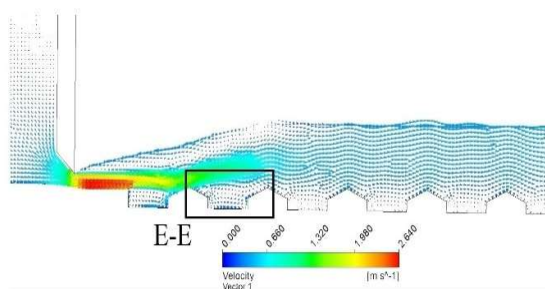
C-C



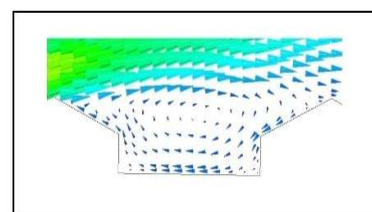
(d)



D-D



(e)



E-E

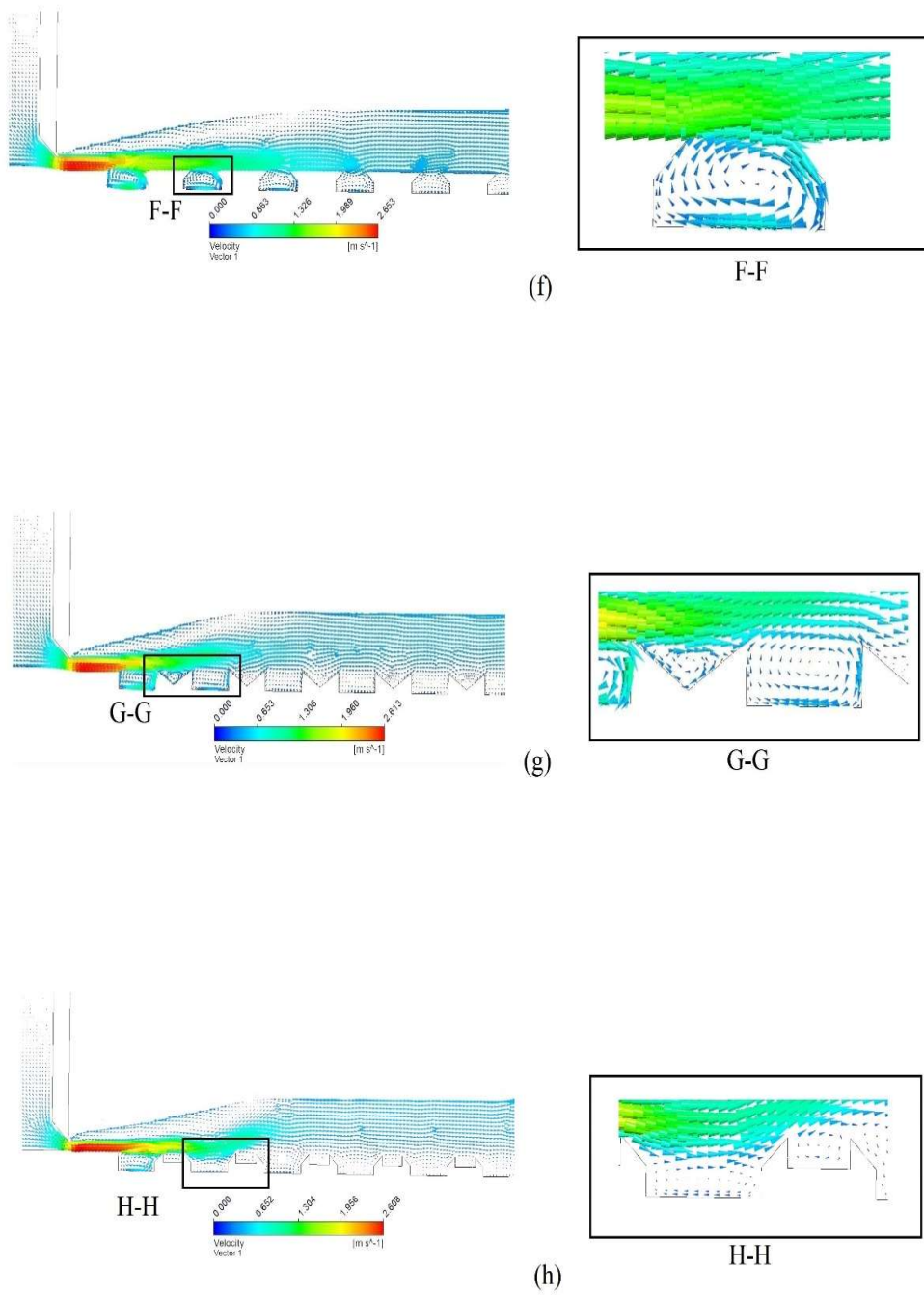
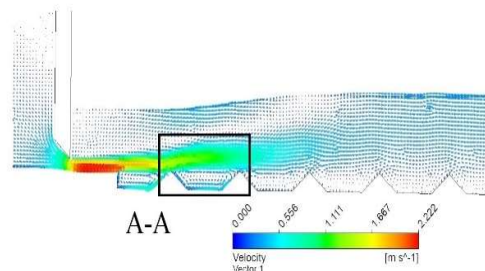
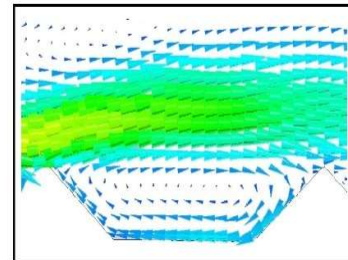


Figure 4.10 Numerically generated velocity vector for each case of free hydraulic jump

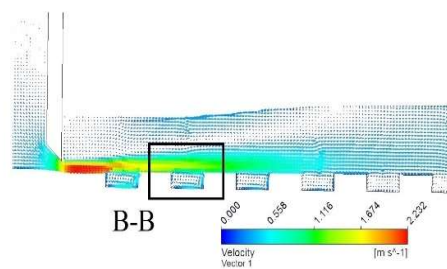
4.3.2.2 Submerged hydraulic jump



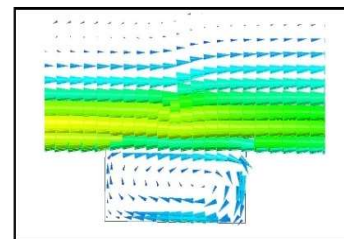
(a)



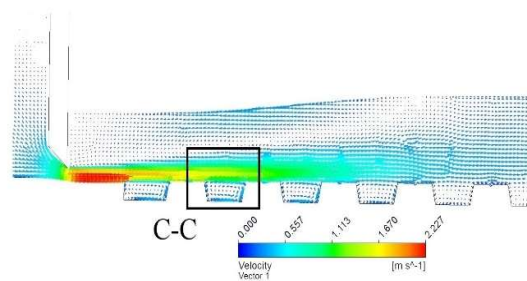
A-A



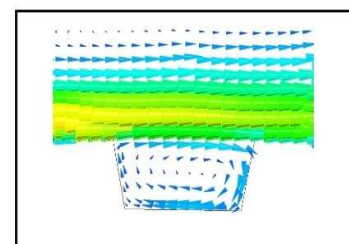
(b)



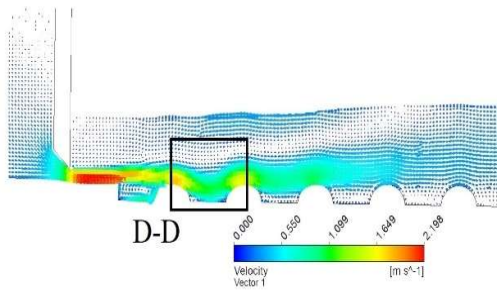
B-B



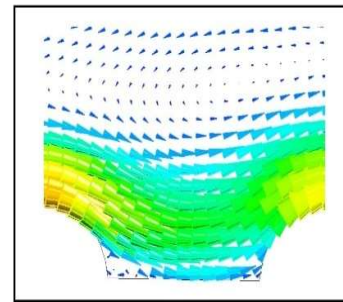
(c)



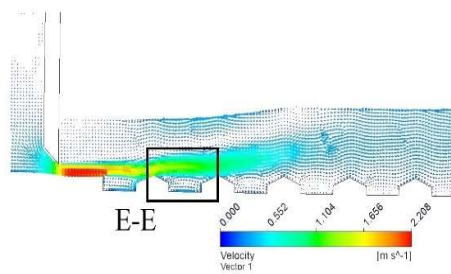
C-C



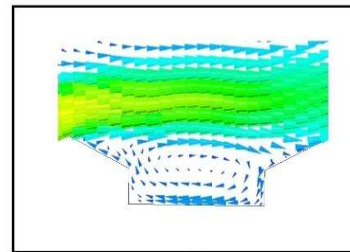
(d)



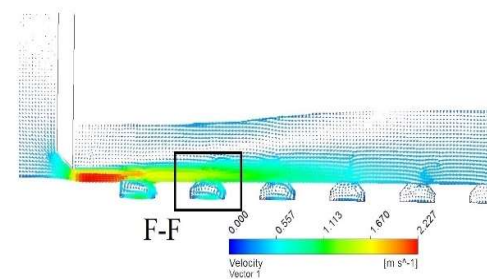
D-D



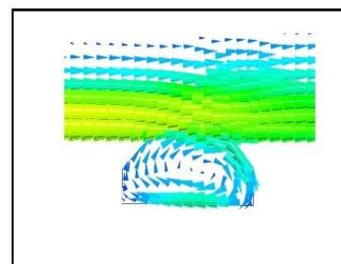
(e)



E-E



(f)



F-F

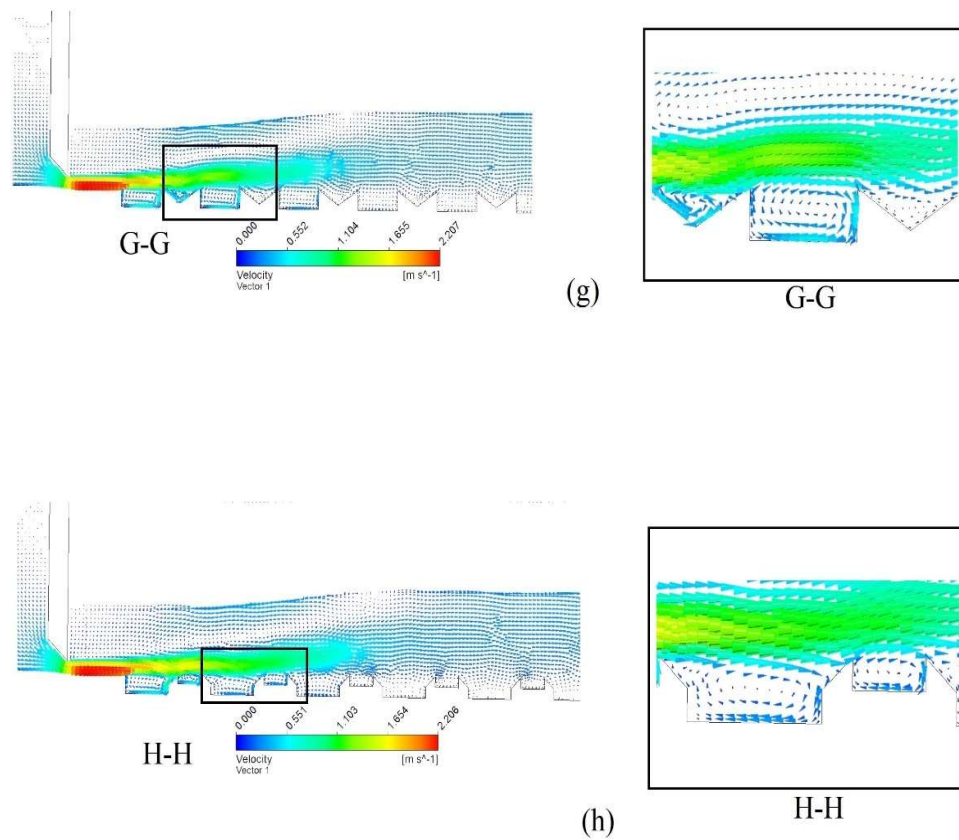


Figure 4.11 (a-h) Numerically generated velocity vector for each case of submerged hydraulic jump

Figure 4.10 & Figure 4.11 illustrate the numerically predicted velocity vectors under the sluice gates on macrorough beds for free and submerged hydraulic jump respectively. It is observed that the flow pattern in the sluice gate with macroroughness for all the models is almost similar. However, the clock wise circulation are noticed in between two consecutive macroroughness elements for most of the shapes and the fluctuation in clockwise circulation are also seen in some models and this may be due to the structural properties of the particular model. Thus, it can be inferred that the circulation of the fluid depends on the shape of the macroroughness.

The circulation of fluid between the two consecutive strip macroroughness reduces the velocity of fluid near the bed and reduces the length of jump. The space between two consecutive strip macroroughness may be termed as cavity region or dead zone as these spaces are occupied with water. The velocities in cavity region are measured numerically and found that the velocities in these regions for most of the models are found approximately tenth part of the mean velocity. This type of justification is valid for both free and submerged hydraulic jump. Such remarks states that the cavity regions are more influencing parameters in case of fast-moving fluid flow for minimizing or dissipating the fluid velocities and to minimize the chance of erosion of downstream channel bed.

4.3.3 Turbulent Kinetic Energy (TKE)

TKE is one of those descriptions that pertain to energy in a turbulent fluid flow situation. Turbulence is another phenomenon in fluid dynamics which is accompanied by unsteady, random, erratic or Zone turbulent flow of fluids where three-dimensional flow velocity, pressure and density are in constant variation. These fluctuations occur on a wide range of spatial and temporal scales. Turbulent kinetic energy is important in various fields of science and engineering, including meteorology, oceanography, aerospace engineering, and civil engineering. Understanding and predicting turbulent flows and their associated kinetic energy are crucial for designing efficient and safe systems, such as aircraft, wind turbines, and pipelines, and for modelling atmospheric and oceanic circulation patterns.

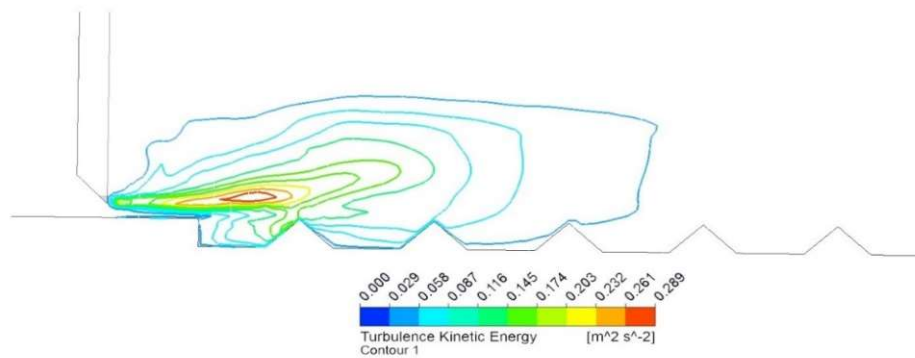
In view of the mathematical definition, TKE is referred to as the mean kinetic energy per unit mass of the turbulent velocity fluctuations and can be written as below.

$$TKE = \frac{1}{2} (\overline{u'^2} + \overline{v'^2} + \overline{w'^2}) \quad (32)$$

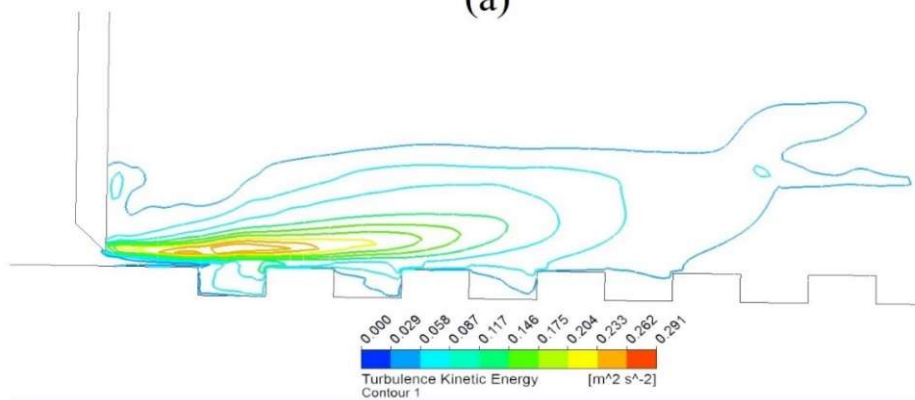
Where, the fluctuating velocity components from the mean velocities in x, y and z- direction are u' , v' , w' respectively. $\overline{u'^2}$, $\overline{v'^2}$, $\overline{w'^2}$ are thought to be the time-averaged of the squared velocity fluctuations which measures the magnitude of turbulence in each direction.

Below sections describe the contours of turbulent kinetic energy over macrorough beds in case of hydraulic jumps.

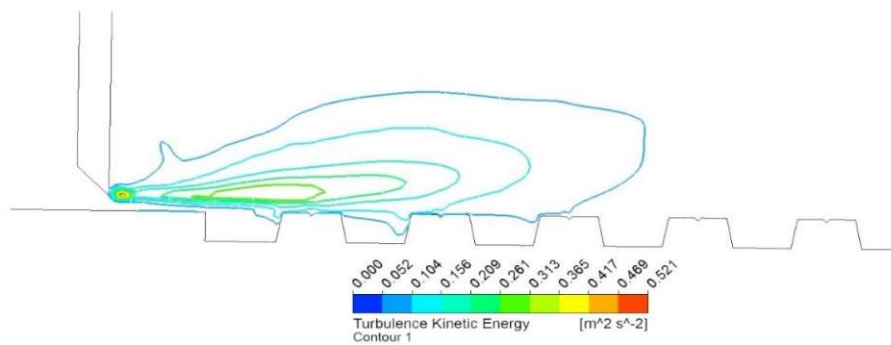
4.3.3.1 Free hydraulic jump



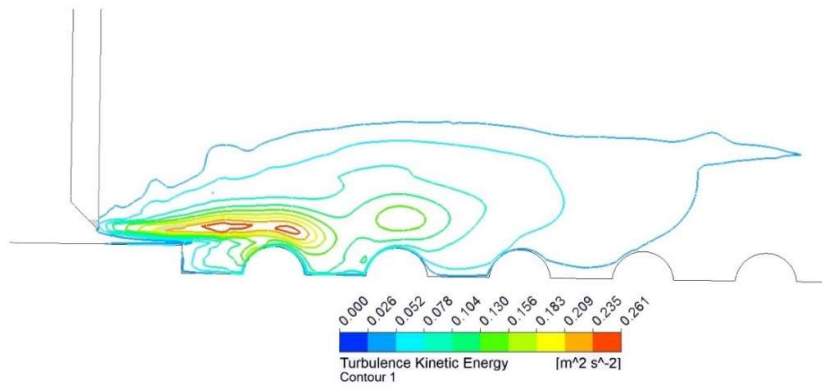
(a)



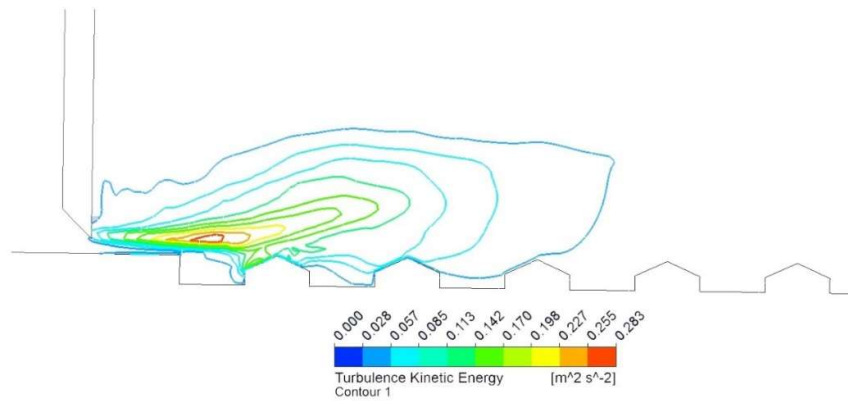
(b)



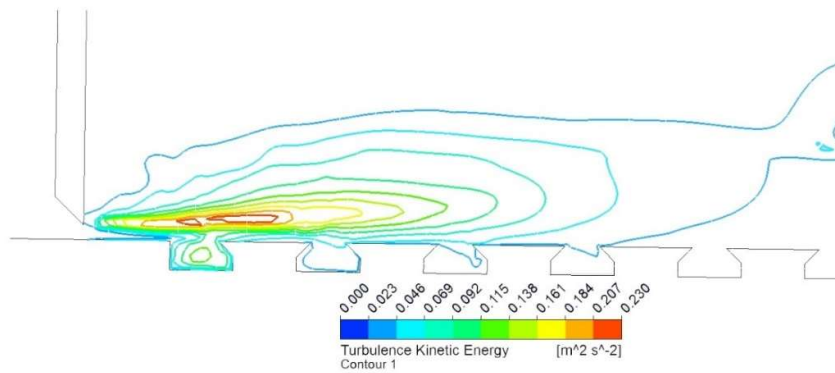
(c)



(d)



(e)



(f)

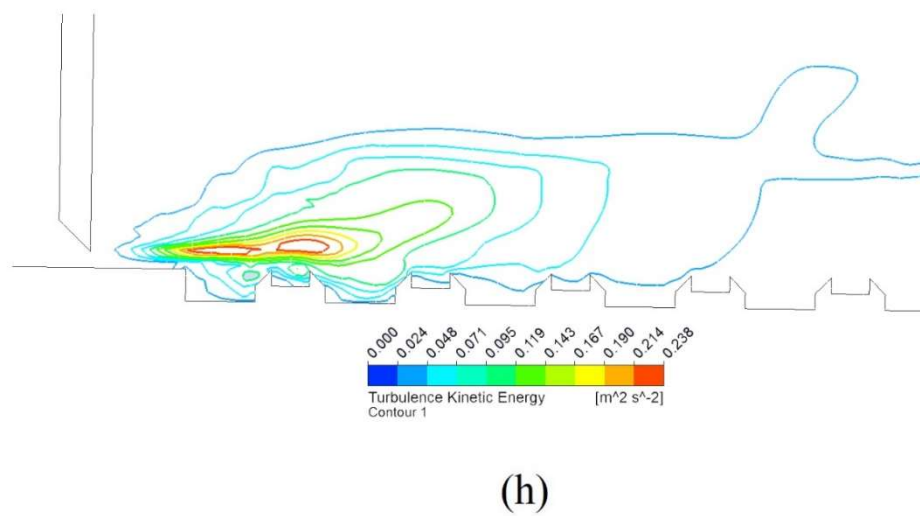
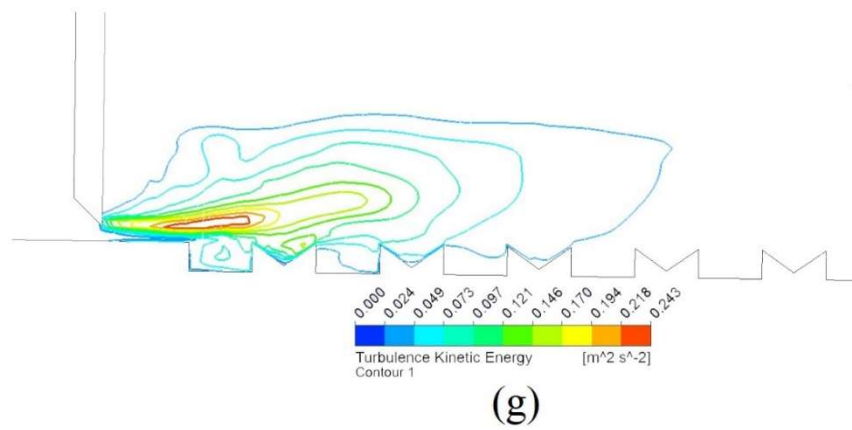
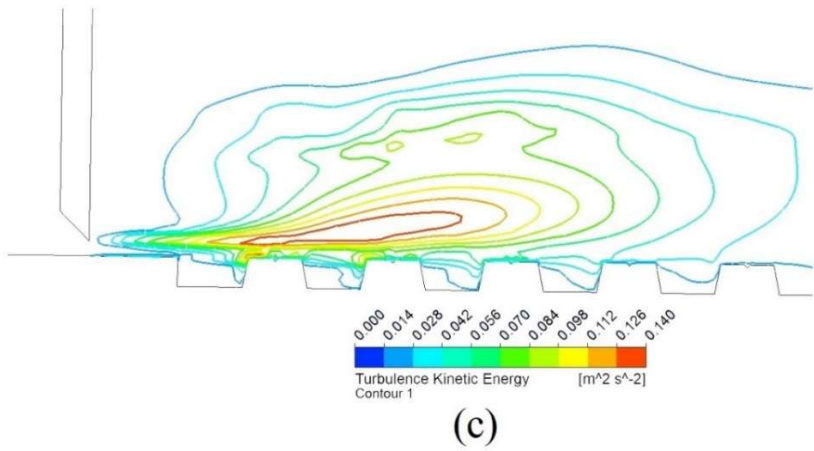
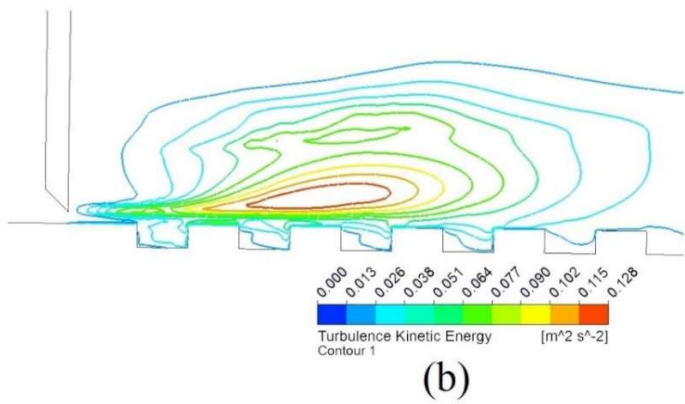
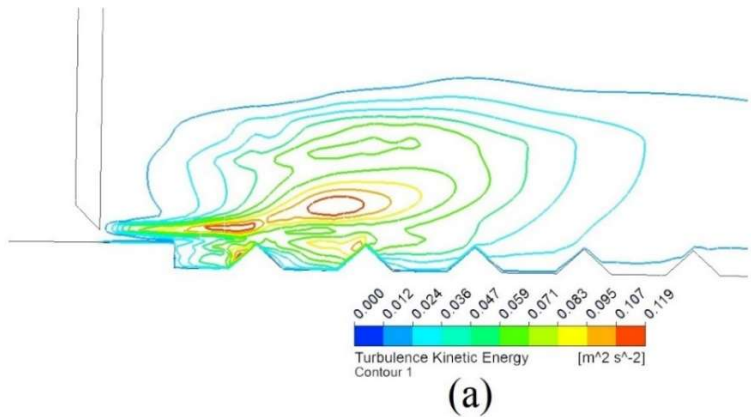
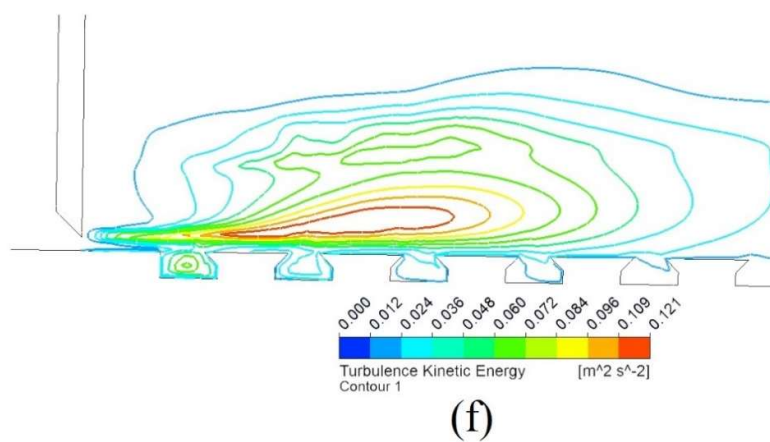
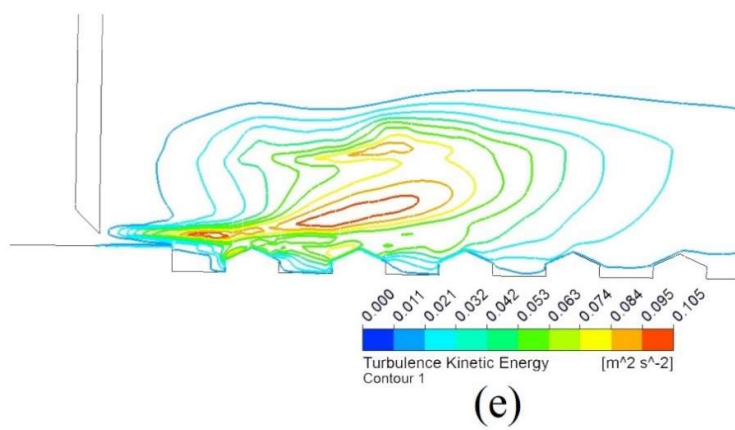
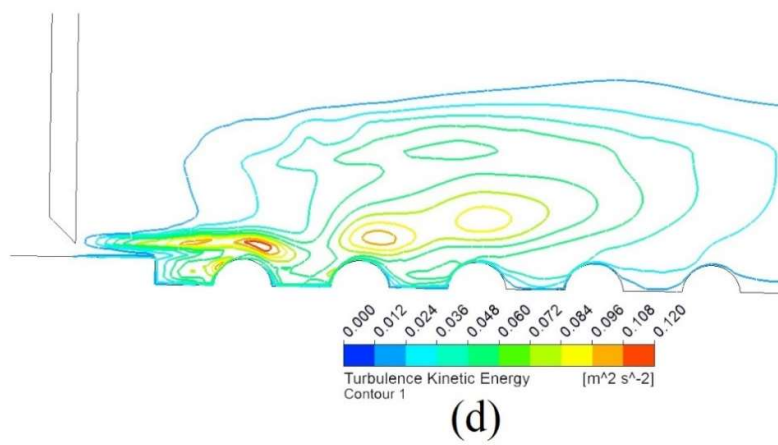
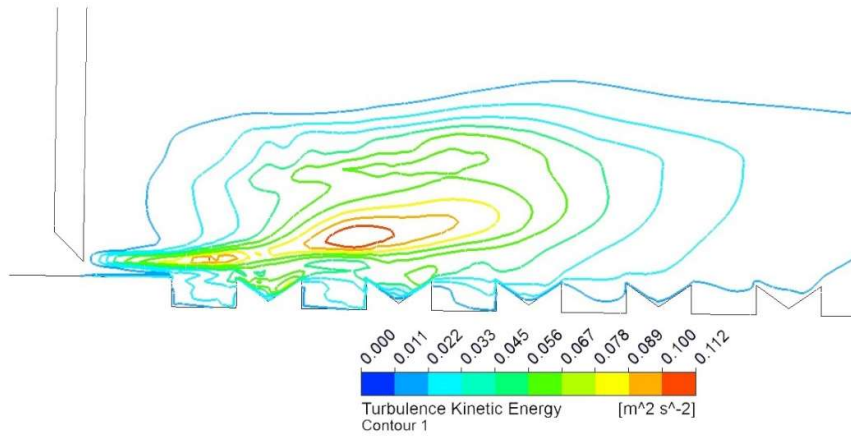


Figure 4.12 (a-h) Numerically generated turbulent kinetic energy contours for each case of free hydraulic jump

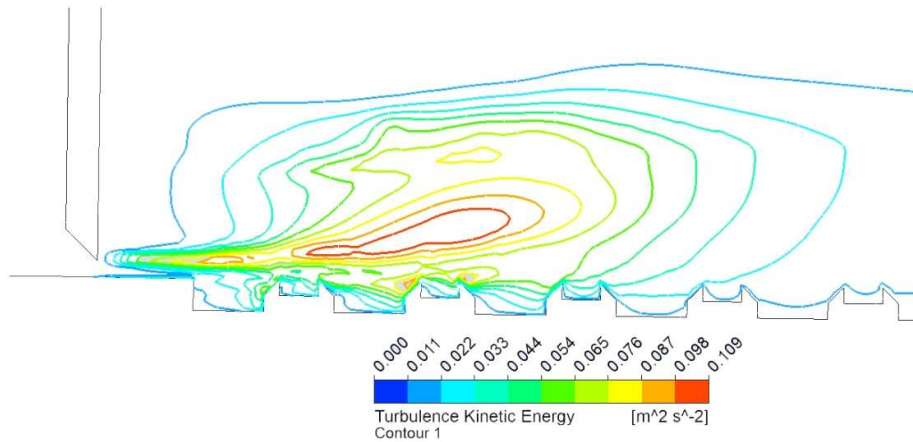
4.3.3.2 Submerged hydraulic jump







(g)



(h)

Figure 4.13 (a-h) Numerically generated turbulent kinetic energy contours for each case of submerged hydraulic jump

Figure 4.12 & Figure 4.13 illustrate the turbulent kinetic energy (TKE) counters for macrorough beds in case of free and submerged hydraulic jump respectively. The turbulence region over macrorough beds within the jump is shown by contours and it can be seen that by providing macroroughness near the bed turbulence kinetic energy reduces by considerable amount. The turbulence region begins near a sluice gate with higher intensity and reduces with the provisions of macroroughness and this is due to

clockwise circulation motion of water in between two consecutive strip macroroughness i.e., cavity region.

4.3.4 Dimensionless Parameters

Dimensionless parameters provide valuable insights into the behaviour of hydraulic jumps and help in the design and analysis of hydraulic structures.

4.3.4.1 Sequent depth ratio (y_2/y_1)

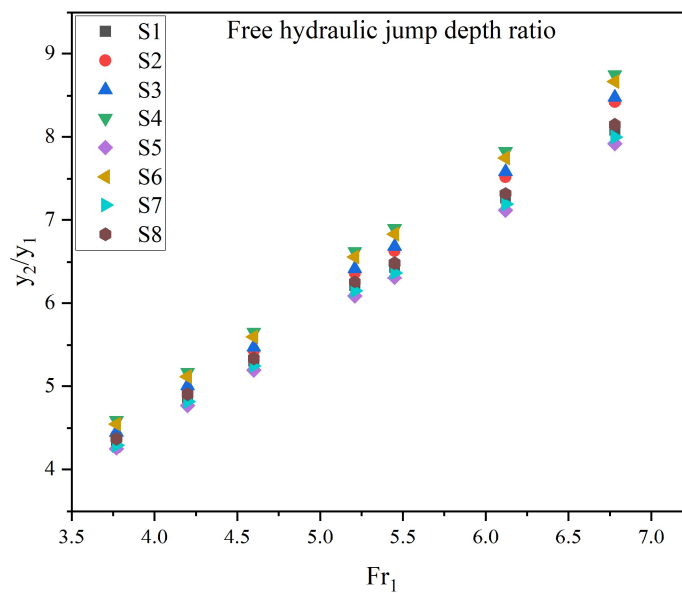


Figure 4.14 Sequent depth ratio vs initial Froude number in case of free hydraulic jump

Comparison of numerically generated results of sequent depth ratio for all eight models is graphically presented in Figure 4.14. From above graph it can be inferred that the model S₁, S₅, S₇ & S₈ yields almost similar results. However, S₁ i.e., triangular model is more effective in enhancing the sequent depth ratio than other one. Quantitatively, it is reported that the sequent depth ratio for S₁ model is 3.13% shorter than S₂ model, 3.95 % shorter than S₃ model, 7.25 % shorter than S₄ model and 6.22 % shorter than S₆ model.

4.3.4.2 Tail water depth ratio (y_4/y_1)

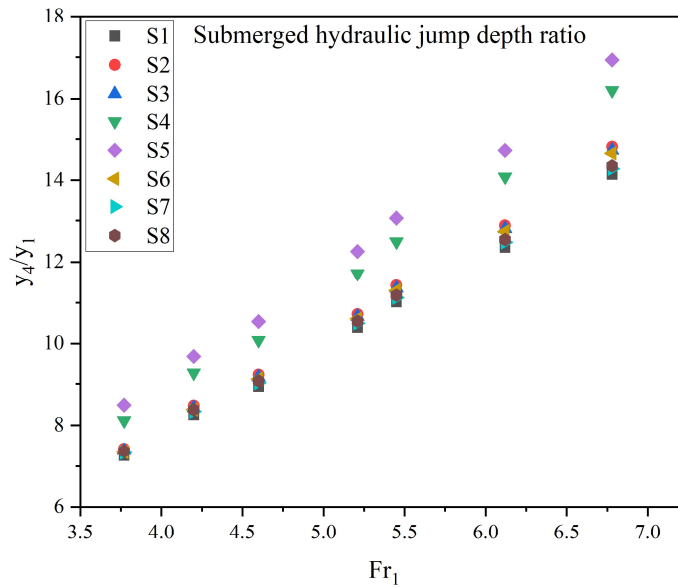


Figure 4.15 Tail water depth ratio versus initial Froude number in case of submerged hydraulic jump

Comparison of numerically generated results of tail-water depth ratio for all eight models are graphically presented in Figure 4.15. From above graph it can be inferred that the model S₁, S₇ & S₈ yields almost similar results. However, S₁ *i.e.*, triangular model is more effective in enhancing the tail-water depth ratio than other one. Quantitatively, it is reported that the tailwater depth ratio for S₁ model is 3.76% shorter than S₂ model, 3.14% shorter than S₃ model, 13.45% shorter than S₄ model, 12.21% shorter than S₅ model and 18.61% shorter than S₆ model.

4.3.4.3 Roller length of jump ratio (L_r/y_1)

Figure 4.16 & Figure 4.17 shows the plot between roller length of jump ratio and initial Froude number in case of free & submerged hydraulic jumps respectively. From Figure 4.16 it can be inferred that the model S₁, S₇ & S₈ has similar results and are more effective in enhancing the roller length of jump ratio than other one. Quantitatively, it is found that the roller length of jump ratio for S₁, S₇ & S₈ model is 5.72% shorter than S₂ or S₃ models and 17.28% shorter than S₄ or S₆ models.

From Figure 4.17 it can be reported that the model S₁, S₇ & S₈ yields almost similar results. Quantitatively, it is found that the roller length of jump ratio for S₁, S₇ & S₈ model is 5.72 % shorter than S₂ or S₃ or S₆ models and 4.37% shorter than S₄ or S₅ models.

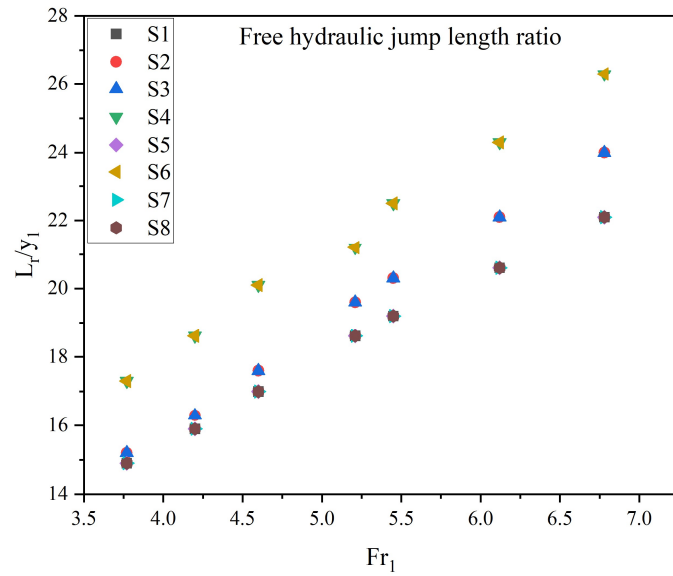


Figure 4.16 length of jump ratio versus initial Froude number in case of free hydraulic jump

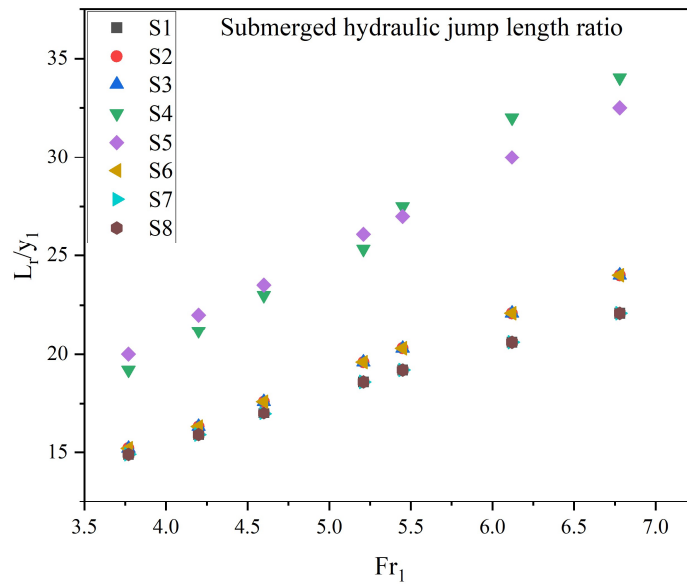


Figure 4.17 length of jump ratio versus initial Froude number in case of submerged hydraulic jump

4.3.4.5 Loss of energy through hydraulic jumps

Energy loss through hydraulic jumps is decided by the difference in specific energy between the pre jump flow states and the post jump flow states. This can be defined more particularly as the relationship between specific energy upstream and specific energy downstream across the jump and expressed in equation for free jump and submerged jump mathematically as follows respectively.

$$E_i = y_i + \frac{u_i^2}{2g} \quad (33)$$

$$E_L = \frac{E_1 - E_2}{E_1} \quad (34)$$

$$E_L = \frac{E_3 - E_4}{E_3} \quad (35)$$

Here, the longitudinal sections within the jump were indicated by the subscript "i" (*i.e.*, $i = 0, 1, 2, 3, 4$). E_L represents relative energy loss through free & submerged hydraulic jump whereas, E_1, E_2 is the specific energy calculated at the upstream and downstream ends of the free hydraulic jump, respectively. Similarly, E_3 and E_4 is the specific energy calculated at the upstream and downstream ends of the submerged hydraulic jump respectively.

Figure 4.18 & Figure 4.19 illustrate the relative energy loss through free & submerged hydraulic jumps. From the results of Figure 4.18 & Figure 4.19 it can be seen that S_1 *i.e.*, triangular model is dissipating more energy than other models.

From Figure 4.18 it can be reported that the relative energy loss in case of free hydraulic jump for S_1 model is 4.87 % higher than S_2 or S_3 model, 6.78% higher than S_4 model, 5.72% higher than S_5 model, 11.04% higher than S_6 model, 3.2% higher than S_7 model & 3.3% higher than S_8 model.

From Figure 4.19 it can be reported that the relative energy loss in case of submerged hydraulic jump for S_1 or S_7 model is 5.43% higher than S_2 or S_3 model, 10.97% higher

than S_4 model, 8.41% higher than S_5 model, 6.31% higher than S_6 model & 1.2% higher than S_8 model.

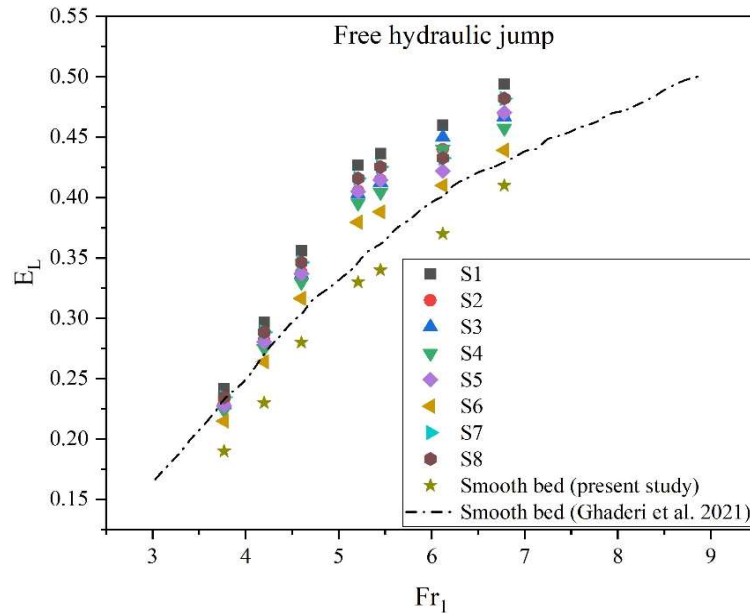


Figure 4.18 Relative energy loss versus initial Froude number in case of free hydraulic jump

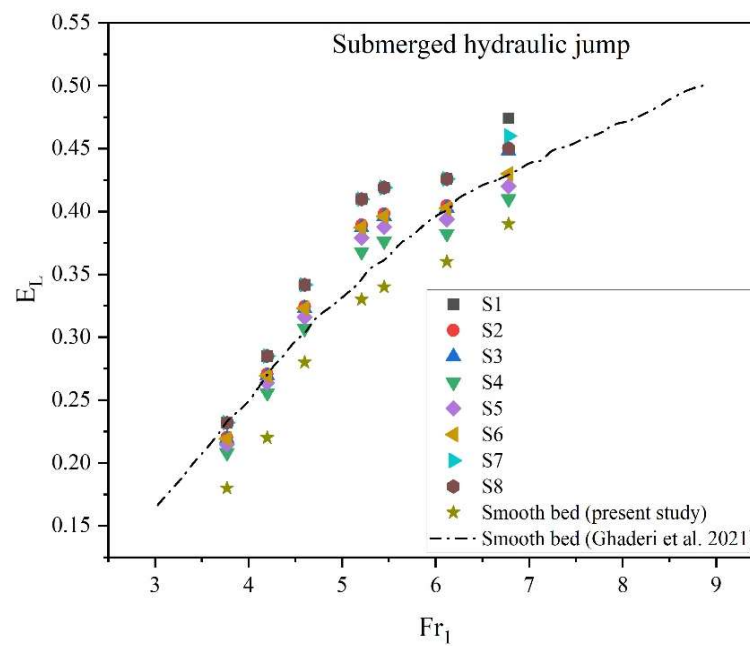


Figure 4.19 Relative energy loss in case of submerged hydraulic jump versus initial Froude number

Earlier research of Ghaderi et al. (2020) [76] shows that a free jump dissipates more energy as compared to the submerged hydraulic jump over macroroughness enhances more energy loss as compared to smooth bed. Findings of the presents study are in well agreement of findings of Ghaderi et al. (2020) [76] & Ghaderi et al. (2021) [18].

4.4 H/B i.e., height to base width ratio of macroroughness

This section deals the effect of H/B *i.e* height to base ratio of macroroughness on free and submerged hydraulic jump. The possible arrangements are also investigated and graphically presented in below sections.

4.4.1 Flow profiles and velocity variation in case of free hydraulic jump

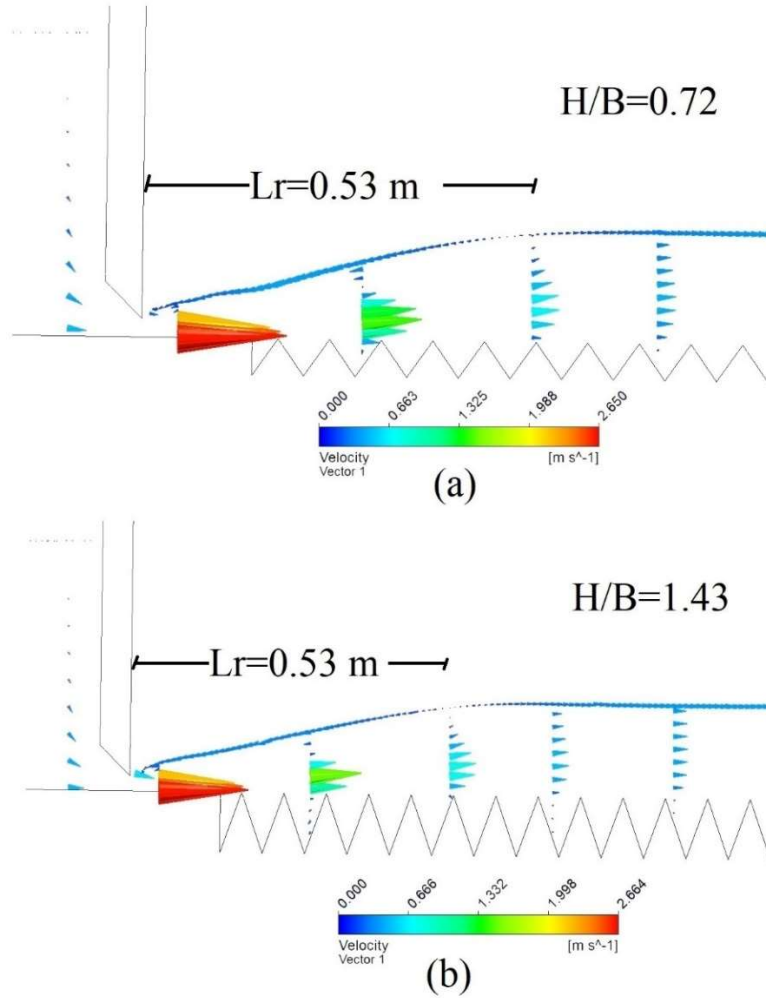


Figure 4.20 (a-b) Numerically generated flow profile of free hydraulic jump for H/B i.e., height to base width ratio of macroroughness

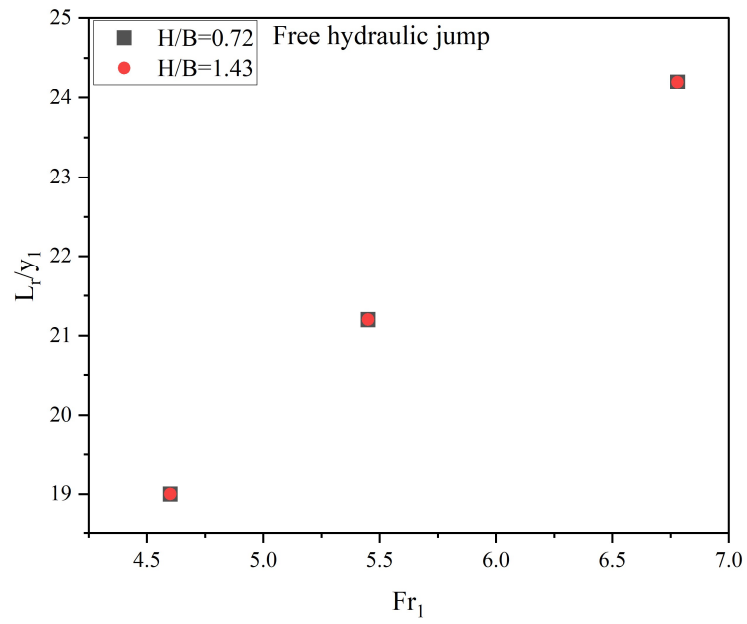


Figure 4.21 Graphical representation of length of jump ratio versus initial Froude number of free hydraulic jump for H/B i.e., height to base width ratio of macroroughness

4.4.2 Flow profiles and velocity variation in case of submerged hydraulic jump

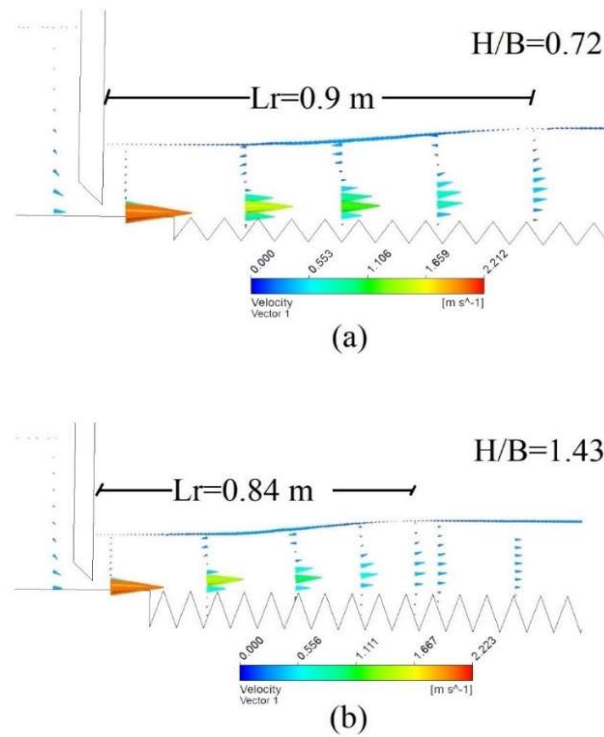


Figure 4.22 (a-b) Numerically generated flow profile with longitudinal velocity variation of submerged hydraulic jump for H/B i.e., height to base width ratio of macroroughness

Figure 4.20 & Figure 4.22 illustrate the pictorial representation of flow profile with longitudinal velocity profiles at different sections of the flume over various H/B ratio of macroroughness's for free and submerged hydraulic jump respectively. Figure 4.20 (a) & Figure 4.22 (a) shows the results of H/B ratio for 0.72 whereas Figure 4.20 (b) & Figure 4.22 (b) shows the result of H/B ratio for 1.43.

To investigate the effect of height to base width ratio on jump characteristics two variations as discussed earlier are considered and the outcomes in the form of roller length of jump ratio is plotted against initial Froude number in Figure 4.21. From Figure 4.21 it can be inferred that on varying the H/B ratio the outcomes are almost same and no significant changes are reported in case of free hydraulic jump. However, from Figure 4.23 the roller length of jump ratio decreases with varying the H/B ratio in case of submerged hydraulic jump. This statement is valid for wide range of Froude

numbers. Quantitatively, the roller length of jump for $H/B = 1.43$ is reported 6.5% shorter than $H/B = 0.72$.

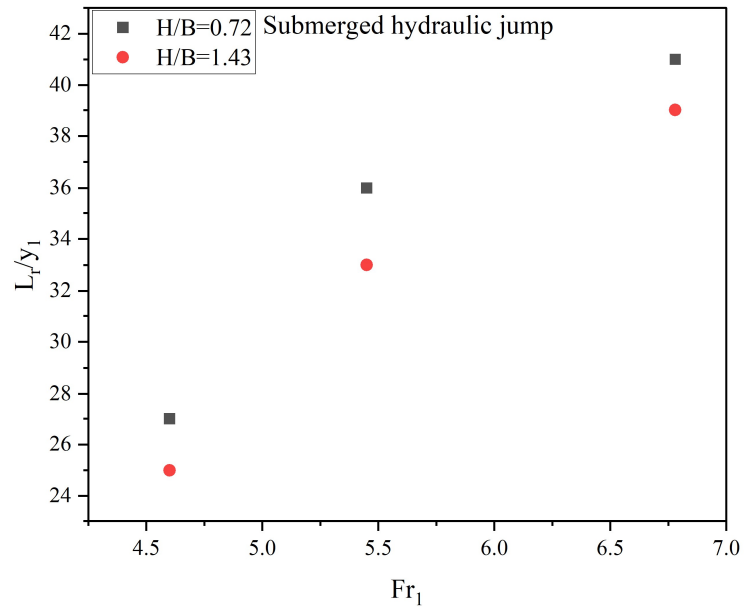


Figure 4.23 Graphical representation of length of jump ratio versus initial Froude number of submerged hydraulic jump for different possible arrangements of H/B i.e., height to base width ratio of macroroughness

4.5 Possible arrangements with H/B ratio

This section explain the effects of various macroroughness on jump characteristics.

4.5.1 Flow profiles and velocity variation in case of free hydraulic jump

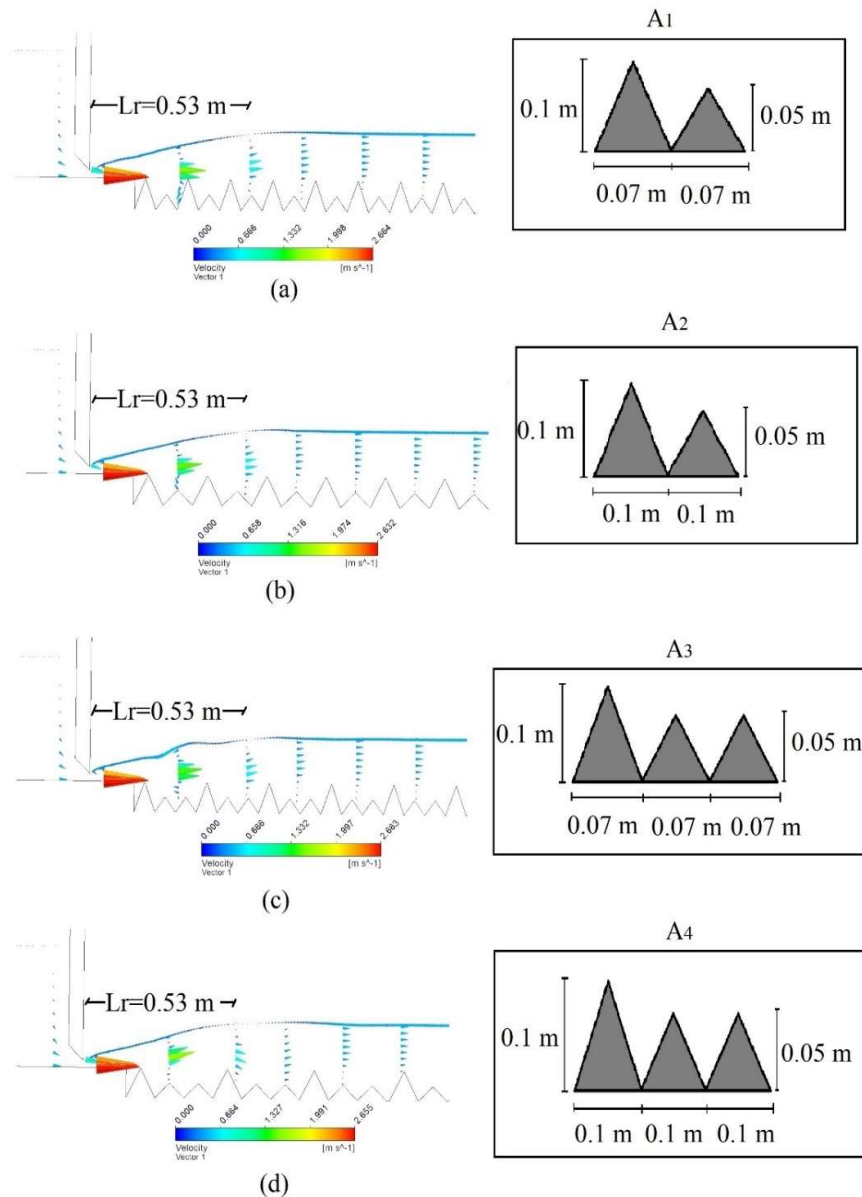
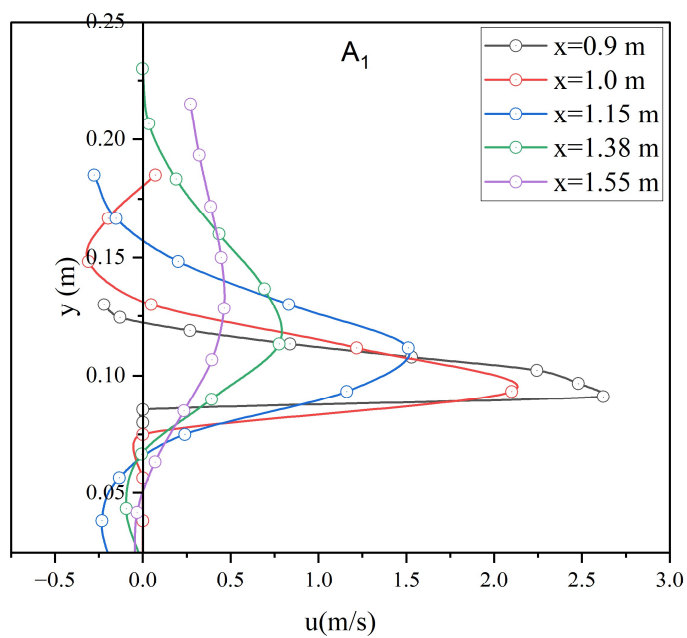
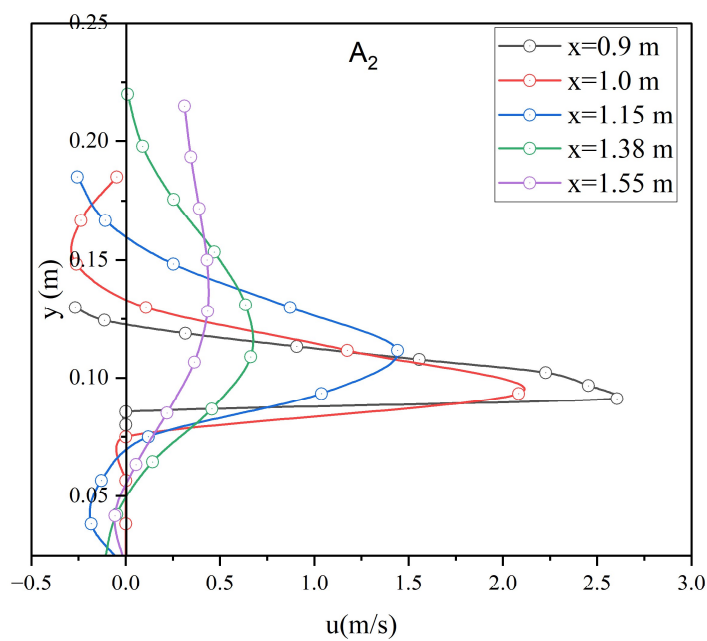


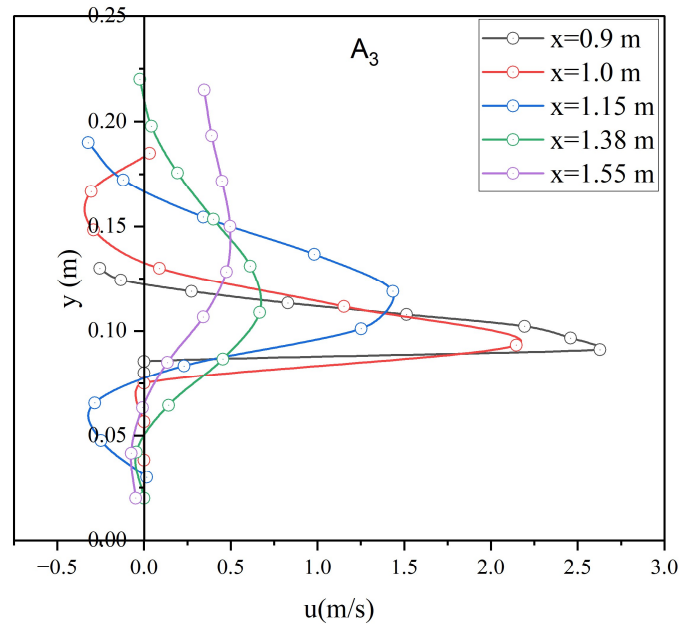
Figure 4.24 (a-d) Numerically generated flow profile with longitudinal velocity variation of free hydraulic jump for different possible arrangements of H/B i.e., height to base width ratio of macroroughness



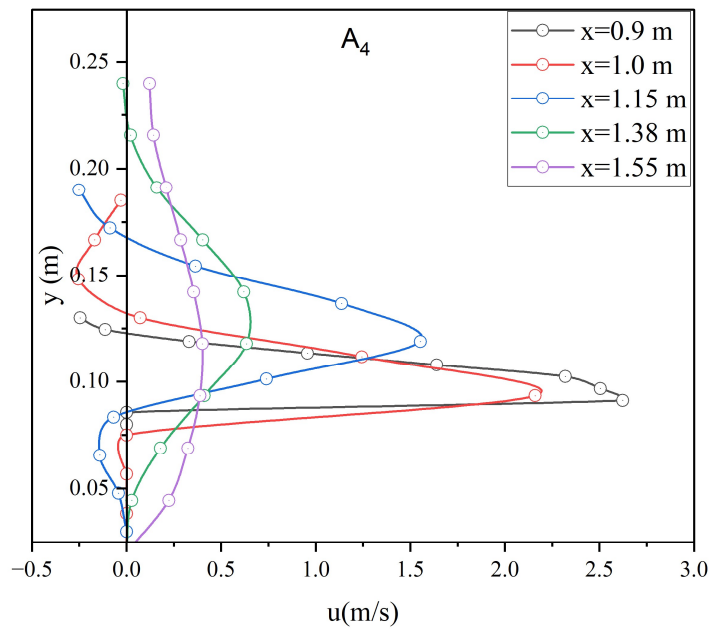
(a)



(b)



(c)



(d)

Figure 4.25 (a-d) Graphical representation of longitudinal velocity variation of free hydraulic jump for different possible arrangements of H/B i.e., height to base width ratio of macroroughness

4.5.2 Flow profiles and velocity variation in case of submerged hydraulic jump

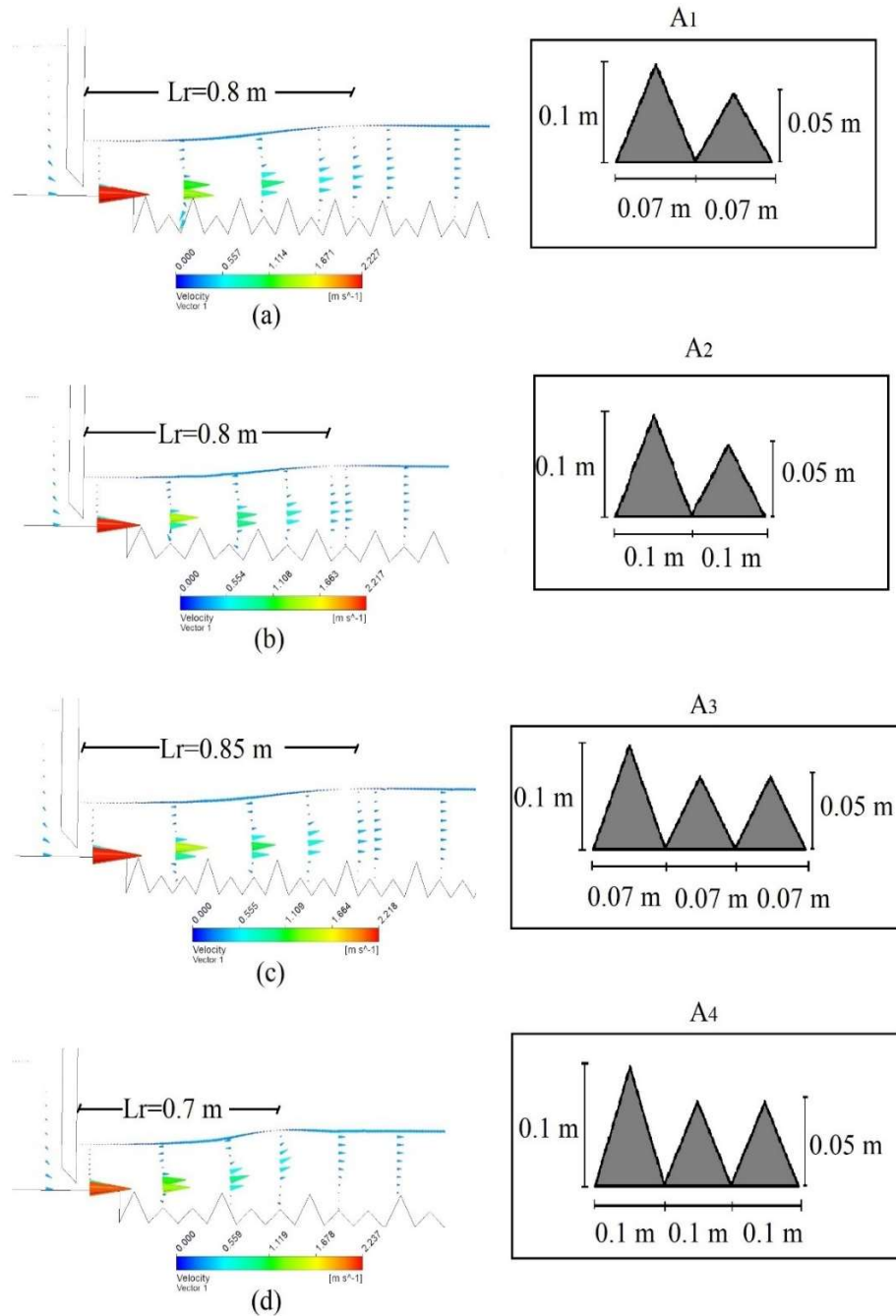
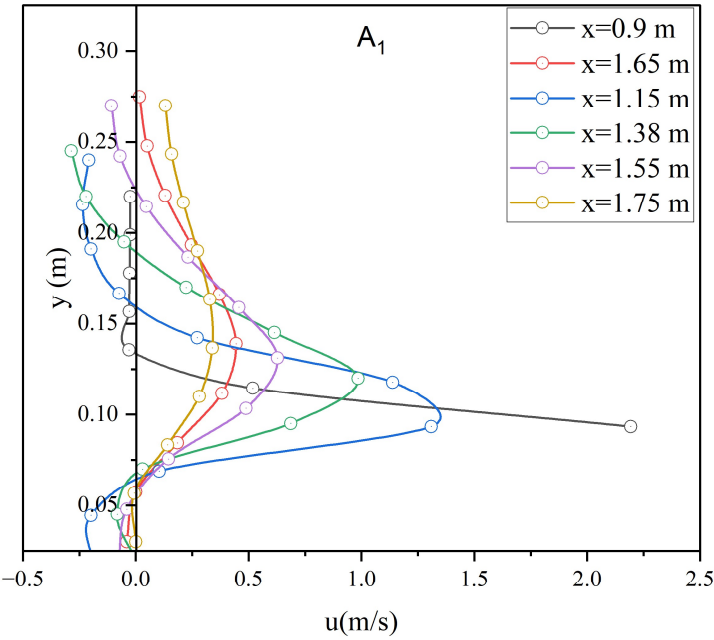
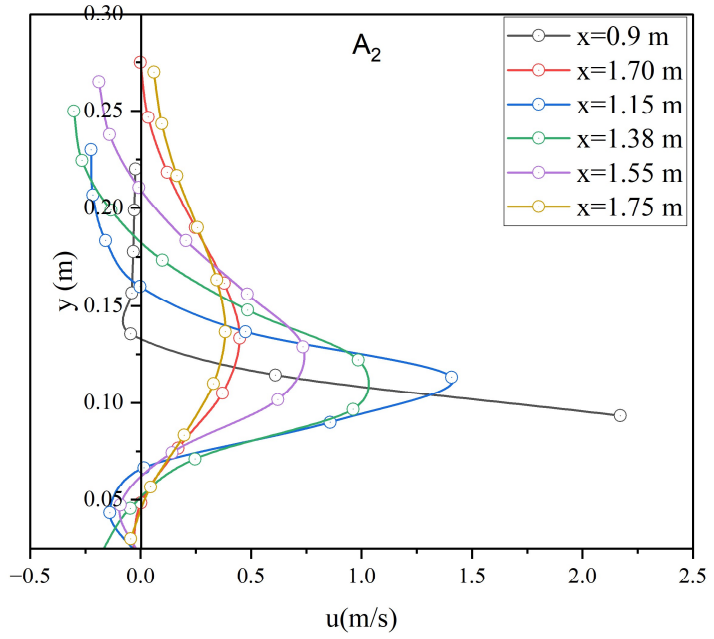


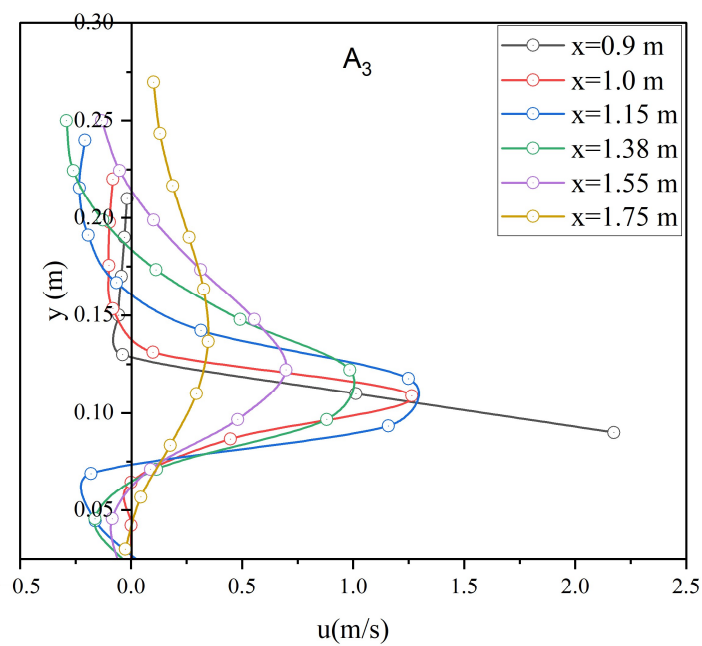
Figure 4.26 (a-d) Numerically generated flow profile with longitudinal velocity variation of submerged hydraulic jump for different possible arrangements of H/B i.e., height to base width ratio of macroroughness



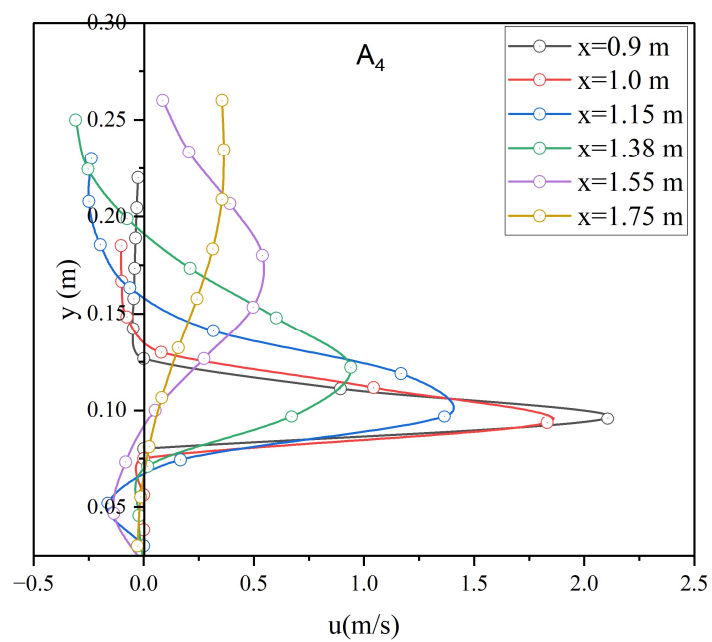
(a)



(b)



(c)



(d)

Figure 4.27 (a-d) Graphical representation of longitudinal velocity variation of submerged hydraulic jump for different possible arrangements of H/B i.e., height to base width ratio of macroroughness

Figure 4.24 & Figure 4.26 illustrate the flow profiles with longitudinal velocities at various sections of the flume including roller length of jump for various arrangement made with the combination of H/B ratio. Here, four combinations *i.e.*, A_1 , A_2 , A_3 , A_4 are prepared using two values of H/B ratio as discussed in earlier sections for free and submerged hydraulic jumps. From Figure 4.25 it is clear that by considering different arrangements the roller length of jump does not change and the flow profile including longitudinal velocities are almost same for all models in case of free hydraulic jump. However, from Figure 4.27 it is clear that on considering different arrangements the roller length of jump changes to some extent in case of submerged hydraulic jump.

4.6 H/I i.e., height to wavelength ratio of macroroughness

4.6.1 Flow profiles and velocity variation in case of free hydraulic jump

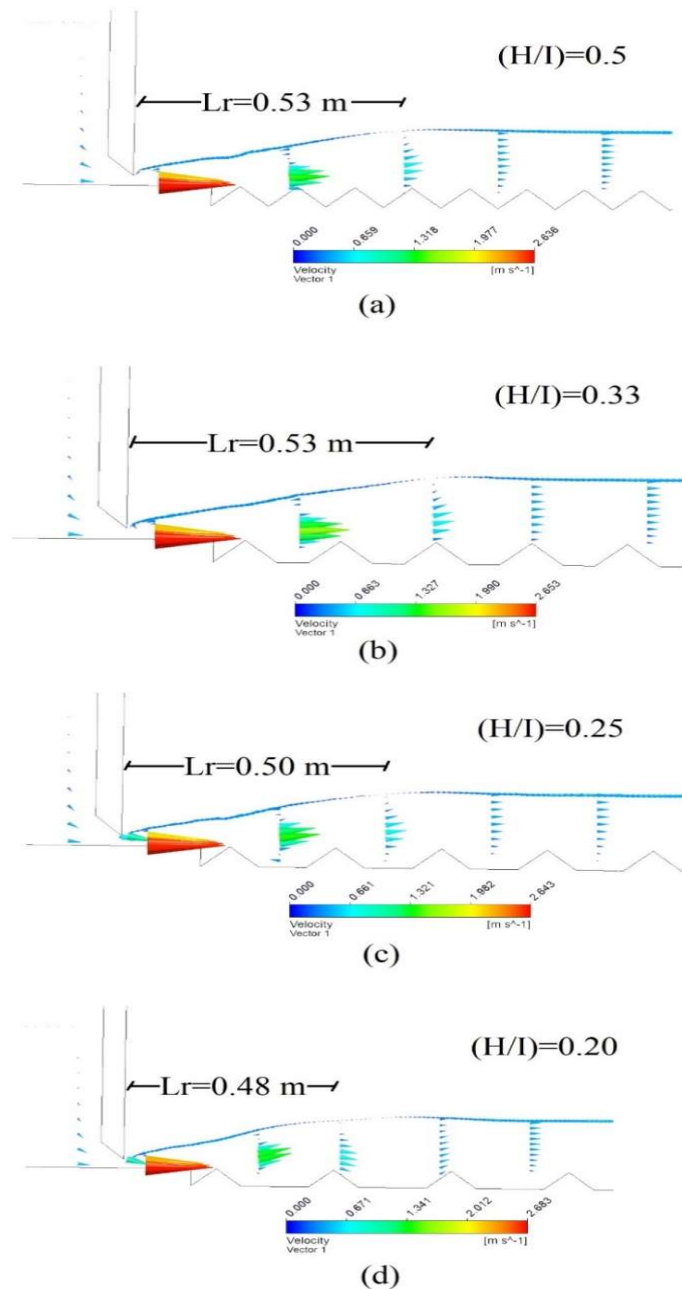


Figure 4.28 (a-d) Numerically generated flow profile with longitudinal velocity variation of free hydraulic jump for H/I i.e., height to wavelength ratio of macroroughness

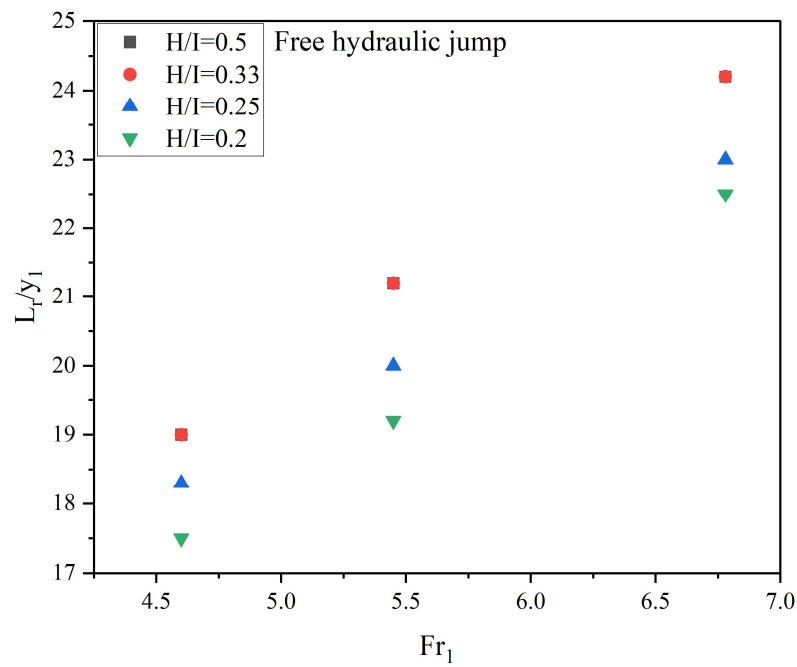
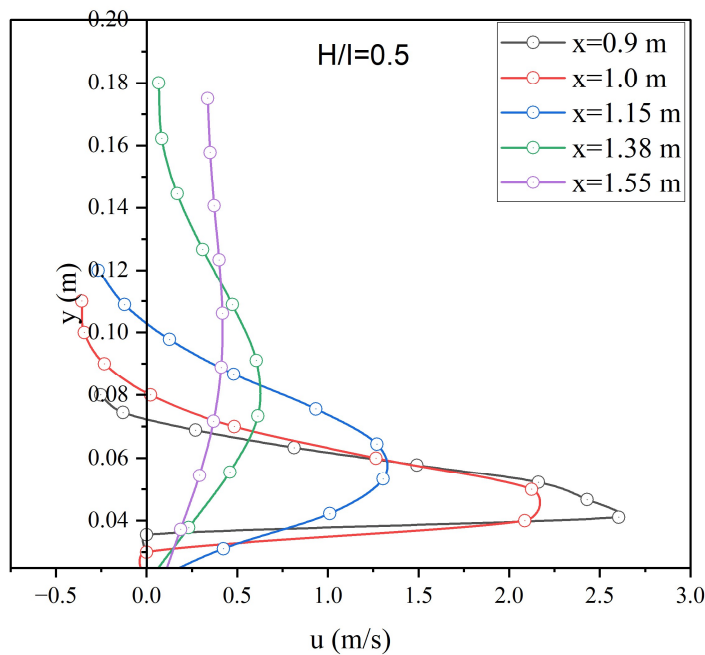
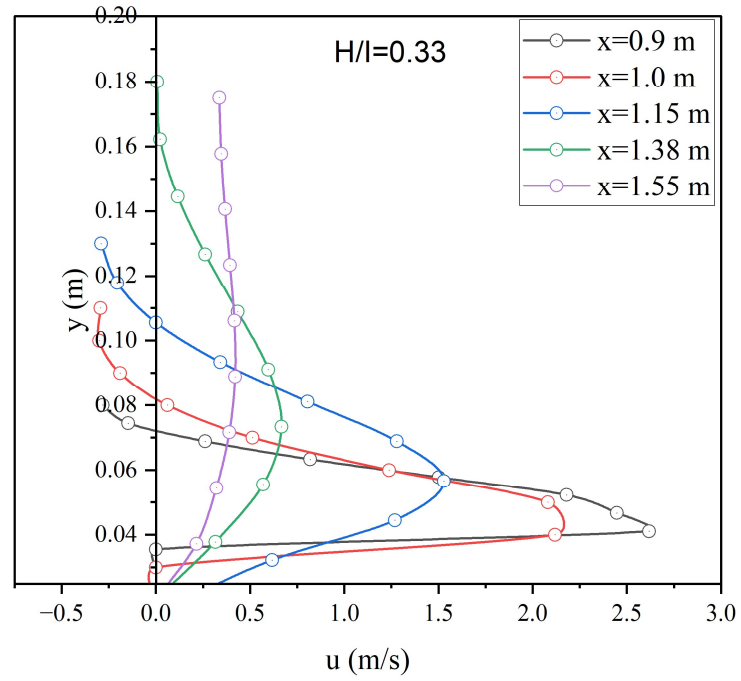


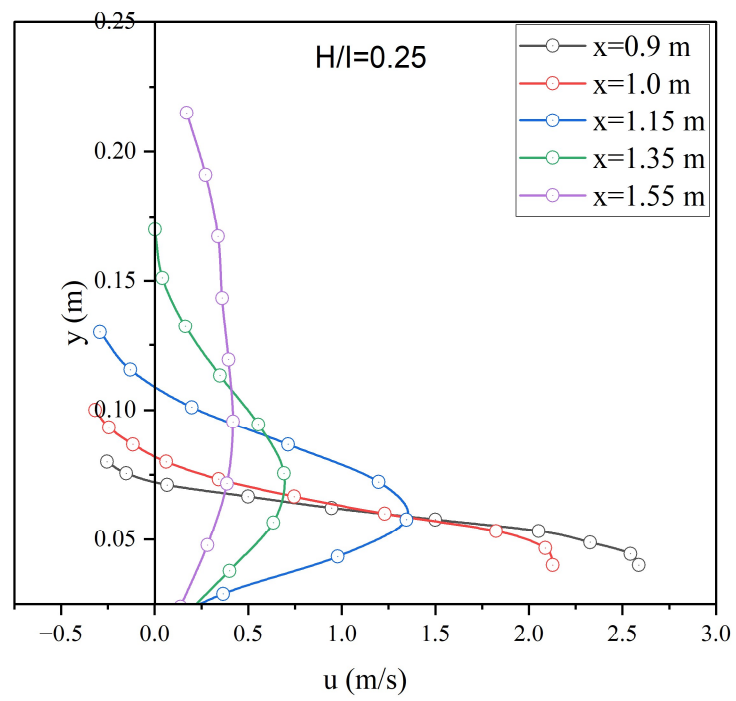
Figure 4.29 Graphical representation of length of jump ratio versus initial Froude number of free hydraulic jump for H/l i.e., height to wavelength ratio of macroroughness



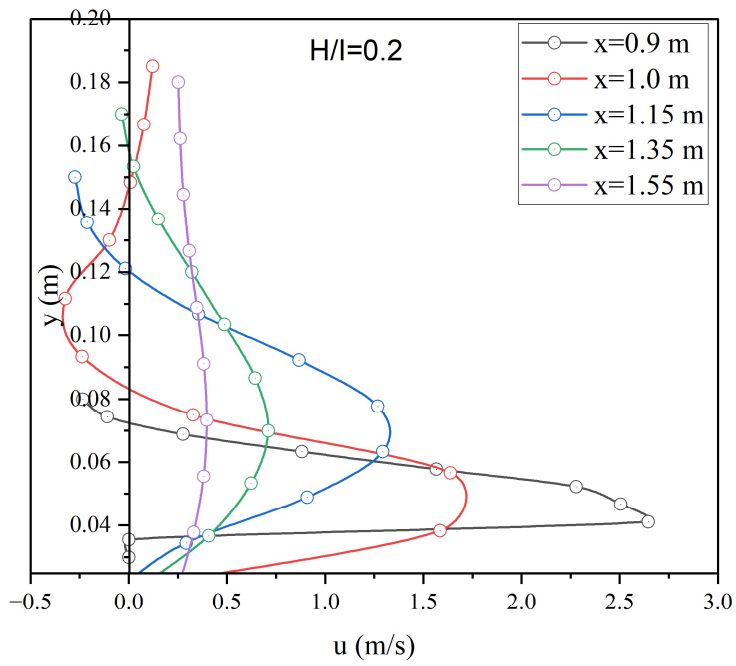
(a)



(b)



(c)



(d)

Figure 4.30 (a-d) Graphical representation of longitudinal velocity variation of free hydraulic jump for H/l i.e., height to wavelength ratio of macroroughness

4.6.2 Flow profiles and velocity variation in case of submerged hydraulic jump

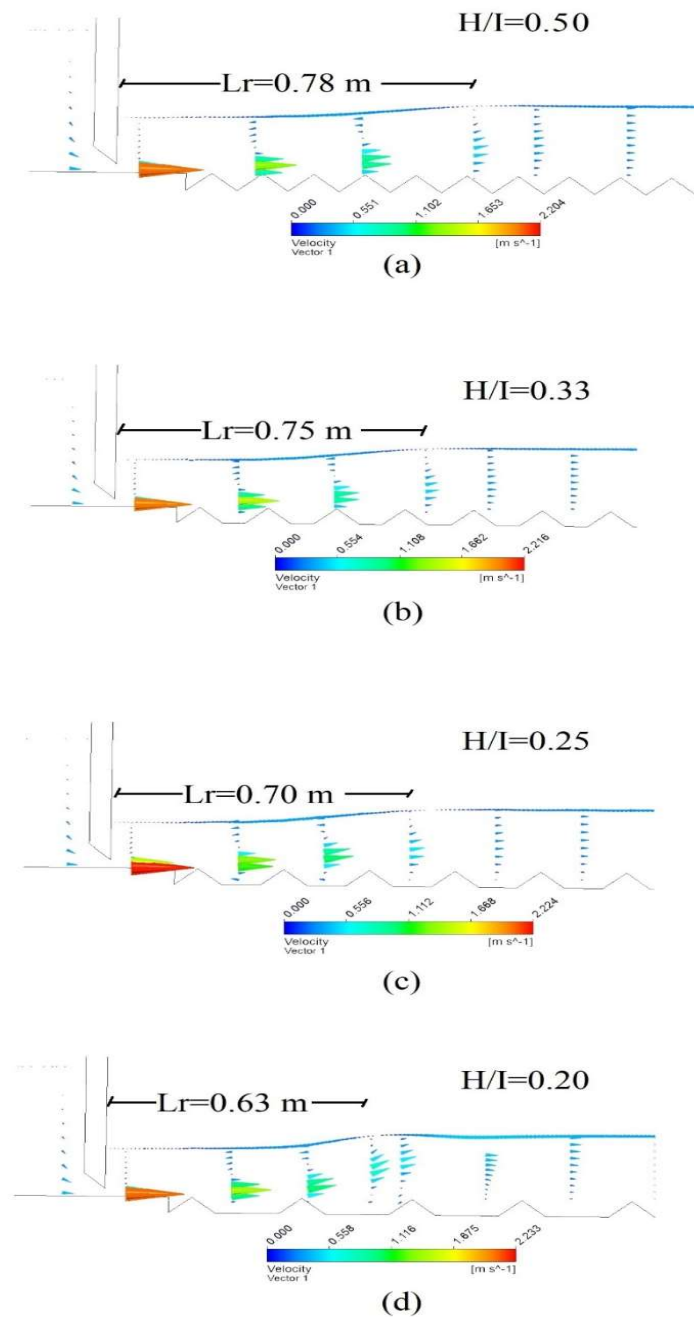


Figure 4.31 (a-d) Numerically generated flow profile with longitudinal velocity variation of submerged hydraulic jump for H/I i.e., height to wavelength ratio of macroroughness

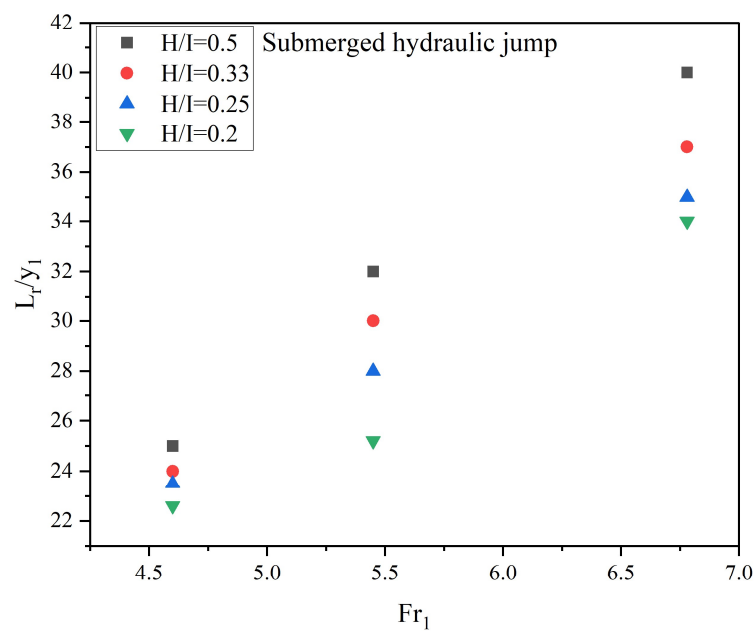
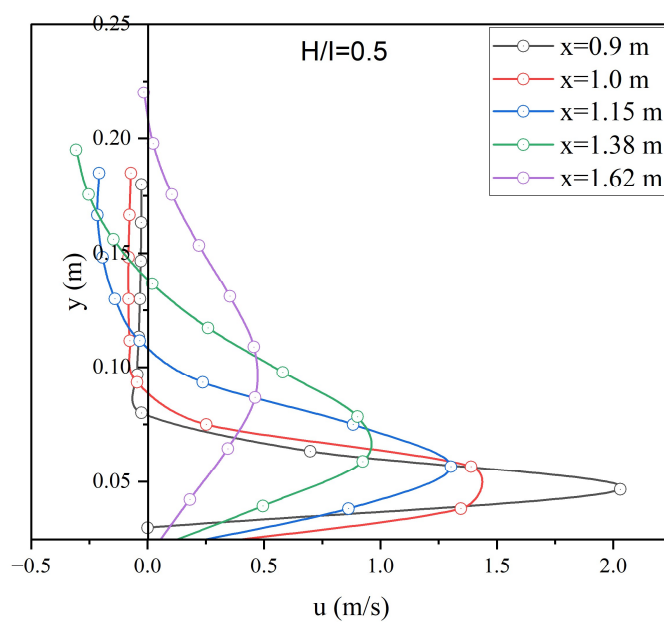
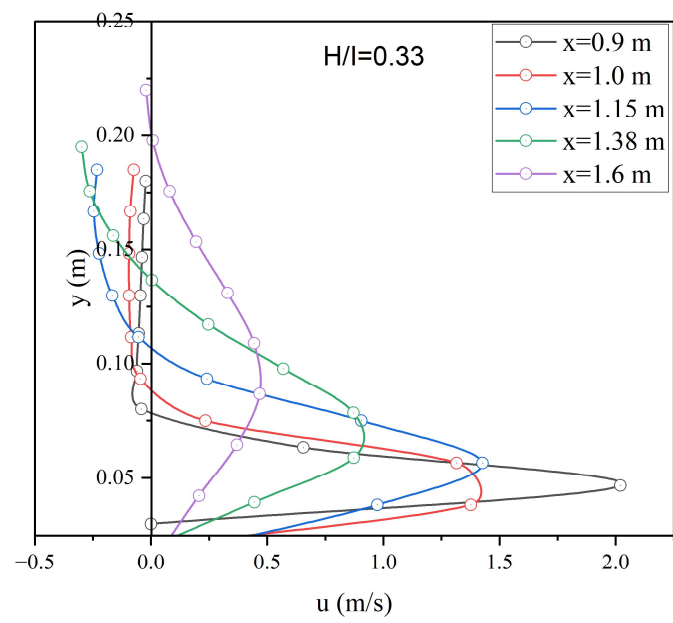


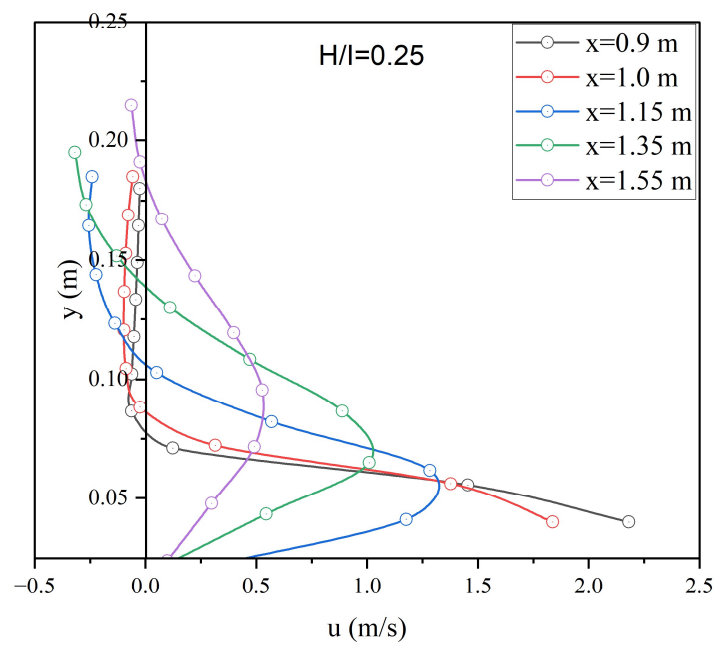
Figure 4.32 Graphical representation of length of jump ratio versus initial Froude number of submerged hydraulic jump for H/I i.e., height to wavelength ratio of macroroughness



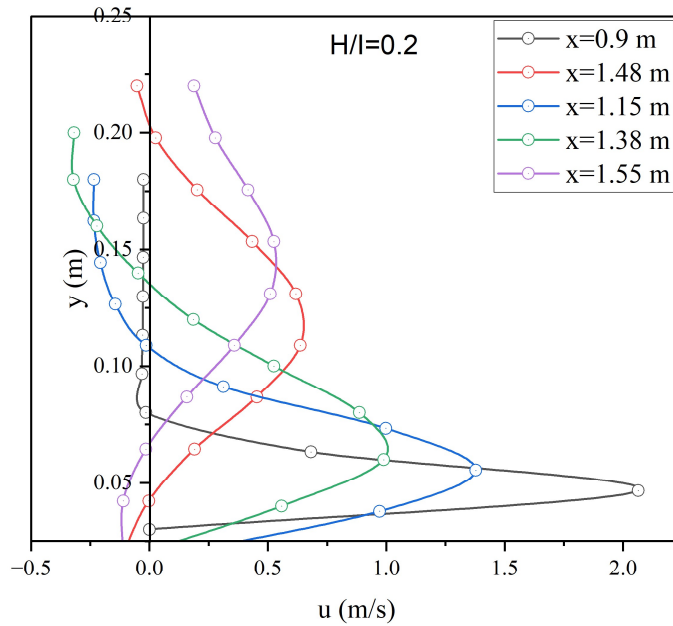
(a)



(b)



(c)



(d)

Figure 4.33 (a-d) Graphical representation of longitudinal velocity profile of submerged hydraulic jump for H/I i.e., height to wavelength ratio of macroroughness. On varying the H/I ratio the spacing between two consecutive strip macroroughness increases and the effect of these are indirectly related to the characteristics of hydraulic jump such as roller length of jump. Figure 4.28 & Figure 4.31 illustrate the pictorial representation of free and submerged hydraulic jump respectively for different H/I ratio i.e., height to wavelength ratio. In Figure 4.28 & Figure 4.31 four values of H/I ratio i.e., 0.5, 0.33, 0.25, 0.2 are considered to investigate the hydraulic jump characteristics.

From the pictorial presentation, it is clear that when the length of the strip macroroughness is more, the velocity distribution function has regained an equilibrium state by the time the flow gets to the subsequent roughness. However for short distance, flow reaches the next macroroughness without sufficient recovery of velocity distribution.

Figure 4.29 & Figure 4.32 represents the dimensionless parameter roller length of jump ratio for free hydraulic jump and for submerged hydraulic jump, respectively. From

Figure 4.29, it can be reported that the $H/I=0.5$ & $H/I=0.33$ has similar results. Whereas, $H/I=0.20$ is capable in reducing roller length of jump than other combination. Quantitatively, roller length of jump ratio for $H/I=0.20$ is 3.38% shorter than $H/I=0.25$, 8.6% shorter than $H/I=0.33$ or $H/I=0.5$.

From Figure 4.32 it can be reported that the $H/I=0.20$ is capable in reducing roller length of jump than other combinations. Quantitatively, roller length of jump ratio for $H/I=0.20$ is 5.56% shorter than $H/I=0.25$, 11.28% shorter than $H/I=0.33$ and 18.8% shorter than $H/I=0.5$.

4.7 Practical challenges in implementing engineered bed features for hydraulic jump control in real-world systems

The present study underscores the effectiveness of strip-type macrorough beds in enhancing hydraulic jump characteristics, notably in improving energy dissipation, reducing jump length, and increasing turbulence for better aeration and mixing. However, practical implementation in real-world river engineering and hydropower systems poses several challenges. Structural integrity and durability are major concerns, as constructing precise geometries like triangular or trapezoidal strips is technically demanding and susceptible to long-term wear from sediment transport, cavitation, and high-velocity flows necessitating the use of high-strength or abrasion-resistant materials. Additionally, closely spaced roughness elements can trap sediment and debris, alter flow behaviour and reduce efficiency, which in turn increases maintenance requirements. Compatibility with existing hydraulic structures may also be limited, particularly in aging infrastructures where retrofitting is constrained by geometry or structural stability.

Moreover, the performance validated under controlled laboratory conditions may vary under real-world flow regimes, including flood events and seasonal discharge changes, thus requiring site-specific customization. Finally, a thorough cost-benefit analysis is crucial to determine the viability of these complex macroroughness shapes compared to traditional energy dissipators like baffle blocks or end sills. Despite these limitations, the study offers valuable design guidance, suggesting that with proper material selection and robust structural design strip-type macrorough elements,

especially triangular forms, hold promise for applications in controlled environments such as hydropower tailraces, irrigation outlets, and urban stormwater channels. Further pilot-scale field studies are essential to validate their long-term performance under variable hydraulic and environmental conditions.

4.8 Engineering strategies for bed roughness design in energy dissipating structures

The design of the channel bed in hydraulic jump type stilling basins plays a crucial role in controlling flow behaviour and enhancing energy dissipation through turbulence generation. This study highlights several key engineering considerations for optimal design. Macroroughness elements, especially strip-type protrusions, effectively modify the post-jump flow field, with triangular strip macroroughness demonstrating superior performance in promoting energy dissipation, roller formation, and turbulence development, particularly under submerged jump conditions. While the height-to-base width ratio (H/B) has minimal impact on jump characteristics, a lower height-to-wavelength ratio (H/I) enhances energy dissipation by increasing flow resistance and interaction zones. The spacing and arrangement of macrorough elements significantly influence flow structure, with varied configurations inducing asymmetric turbulence beneficial for submerged jumps.

Elements oriented perpendicular to the flow generates maximum resistance, strong wake zones, and facilitate flow deceleration and reattachment, making this orientation preferable in standard rectangular channels. For numerical simulations, the RNG $k-\epsilon$ turbulence model provides the most reliable predictions of turbulence intensity, energy loss, and jump length over rough beds. Additionally, omitting the upstream reservoir in simulations can halve computational effort without affecting jump formation when the focus is on post-jump behaviour. In summary, careful consideration of macroroughness geometry, spacing, orientation, arrangement patterns, and appropriate turbulence modelling ensures efficient stilling basin design, reduces erosion risk, and improves hydraulic jump stability across various flow regimes.

CHAPTER 5: CONCLUSION, FUTURE SCOPE AND SOCIAL IMPACT

Overall, this research aimed to enhance the understanding of free and submerged hydraulic jumps by exploring various flow characteristics such as sequent depth ratio, tailwater depth ratio, roller length of jump ratio, relative energy loss including longitudinal velocity profile, and flow pattern in the cavity region over different shapes of macroroughness. Based on the numerical investigation, the key findings of the present study are summarised below:

To predict the sequent depth ratio of a hydraulic jump using Computational Fluid Dynamics (CFD), four different turbulence models such as standard $k-\varepsilon$, realizable $k-\varepsilon$, RNG $k-\varepsilon$, and SST $k-\omega$ were compared for coarse to fine mesh element sizes i.e., 50 mm, 30 mm, 20 mm, 10 mm, 9 mm, and 7 mm. It is noted that relative error in predicting the sequent depth ratio reduces with reduced mesh element size. Further, the sequent depth ratio was predicted with an accuracy of more than 98% using RNG $k-\varepsilon$ model followed by realizable $k-\varepsilon$, SST $k-\omega$ and standard $k-\varepsilon$ model for 9 mm mesh element size. Therefore, it can be stated that the complex fluid flow phenomenon such as hydraulic jump can be well investigated by CFD using RNG $k-\varepsilon$ turbulence model at finer mesh.

Four flow domains with two different approaches for the analysis of classical hydraulic jump were studied and found that the jump characteristics such as flow profile, and longitudinal velocity profiles do not change by applying different approaches on different flow domains as discussed in the present study. However, the computational time can be saved up to 50 % by reducing the flow domain.

Eight geometrically different strip macroroughness namely triangular, rectangular, trapezoidal, semicircular, and four irregular shapes were used to investigate the effect of shape on the characteristics of hydraulic jump such as sequent depth ratio, tailwater depth ratio, roller length of jump ratio, and relative energy loss, longitudinal velocity profile. The result showed that the average value of the length of the roller for trapezoidal and triangular macroroughness is found to be 23 % and 46.06 % shorter

than the smooth bed in the case of a free hydraulic jump. However, these values reduce to 17.85 % and 36.33 % in the case of submerged hydraulic jumps. It is also reported that the triangular model is more capable of reducing the sequent depth ratio up to 7.25%, tailwater depth ratio up to 18.61 %, and increasing the relative energy loss up to 11.04 % as compared to smooth bed. This implies that the macroroughness is effective in enhancing the jump characteristics downstream of the hydraulic structure.

Velocity vector variations are plotted for all eight different macroroughness in which clockwise circulation was noticed in the cavity regions. The magnitude and direction of circulation of fluid mainly depend on the shape of macroroughness. The reduction in the roller length of the jump may be due to clockwise circulation. Thus, a properly designed macroroughness can improve the jump characteristics and maximize the relative energy loss.

Two values of the height-to-base width ratio *i.e.*, 0.72 and 1.43 were considered to determine its effect on the roller length of jump ratio. Negligible variation in roller length of jump ratio was reported in the case of free hydraulic jump whereas roller length of jump ratio reduced up to 7 % on increasing the height to base width ratio in the case of submerged hydraulic jump. This signifies that on varying the height to base width ratio the roller length of jump ratio reduces for submerged hydraulic jump.

Four different models having values of the height-to-wavelength ratio *i.e.*, 0.2, 0.25, 0.33, and 0.5 were considered to study its effect on the roller length of the jump. Results showed that the model having a ratio equal to 0.20 was capable of reducing the roller length of jump ratio up to 8.6 % in the case of free jump and 18.8 % in the case of submerged jump when compared with other models. This signifies that on increasing the spacing between two consecutive macroroughness the roller length of the jump reduces.

In the future, macrorough beds may be used to reduce sediment transport and prevent scouring downstream of hydraulic structures. The study and application of hydraulic jumps on macrorough beds using CFD have significant implications for water management and hydraulic engineering. Ongoing research and interdisciplinary collaboration will be critical in realizing these benefits and overcoming the challenges

associated with their implementation. Computational Fluid Dynamics (CFD) has a wide range of social impacts in a variety of industries, including environmental sustainability, public safety, and healthcare. CFD eliminates the need for physical prototypes and extensive trial and error testing, saving time and money in product development across a variety of industries.

REFERENCES

- [1] Novak, P., Moffat, A.I.B., Nalluri, C. and Narayanan, R.A.I.B., 2017. Hydraulic structures. *CRC Press*.
- [2] Federal Emergency Management Agency, USA. “Why Dams Fail”. In: (2017). url: <https://www.fema.gov/why-dams-fail>.
- [3] Peterka, A.J., Bradley, J.N., Robertson, J.M., Harleman, D.R. and Hubbard, P.G., 1958. Discussion of “Turbulence Characteristics of the Hydraulic Jump”. *Journal of the Hydraulics Division*, 84(6), pp.45-58.
- [4] Bureau of Indian Standards Code 4997 (1968), “Criteria for design of hydraulic jump type stilling basin with Horizontal and sloping apron”. *Bureau of Indian Standards, New Delhi, India*
- [5] Hager, Willi H. 1988. “Ressaut Hydraulique Du Type b Dans Un Canal a Forte Pente.” *Journal of Hydraulic Research* 26(5): 539–58.
- [6] Te Chow, V., 1959. Open channel hydraulics.
- [7] Wiggert, J.M., Erfle, P.D. and Morris, H.M., 1971. Roughness elements as energy dissipators of free-surface flow in circular pipes. *Highway Research Rec*, pp.373-64.
- [8] Sarikelle, S. and Simon, A.L., 1980. Field and Laboratory Evaluation of Energy Dissipators for Culvert and Storm Drain Outlets. Department of Civil Engineering, University of Akron.
- [9] Wiggert, J.M. and Erfle, P.D., 1972. Culvert velocity reduction by internal energy dissipators. CONCRETE PIPE NEWS, VOL 24, NO 5, P 87-94, OCT 1972. 10 FIG, 4 TAB.
- [10] Korom, S.F., Sarikelle, S. and Simon, A.L., 1990. Design of hydraulic jump chambers. *Journal of irrigation and drainage engineering*, 116(2), pp.143-153.
- [11] Knight, D.W., 1996. River channel and floodplain hydraulics, Chapter

5. Floodplain processes, pp.139-181.

[12] Rajaratnam, N., 1967. Hydraulic jumps. In *Advances in hydrosience* (Vol. 4, pp. 197-280). *Elsevier*.

[13] McCorquodale, J.A., 1986. Hydraulic jumps and internal flows. *Encyclopedia of fluid mechanics*, 2, pp.120-173.

[14] Hager, W.H. and Hager, W.H., 1992. Classical hydraulic jump. Energy dissipators and hydraulic jump, pp.5-40.

[15] De Padova, D. and Mossa, M., 2021. Hydraulic jump: a brief history and research challenges. *Water*, 13(13), p.1733.

[16] Chanson, H., 2004. *Hydraulics of open channel flow*. *Elsevier*.

[17] Murzyn, Frédéric, and Hubert Chanson. 2009. "Free-Surface Fluctuations in Hydraulic Jumps: Experimental Observations." *Experimental Thermal and Fluid Science* 33(7): 1055–64. <http://dx.doi.org/10.1016/j.expthermflusci.2009.06.003>.

[18] Ghaderi, Amir, Mehdi Dasineh, Francesco Aristodemo, and Costanza Aricò. 2021. "Numerical Simulations of the Flow Field of a Submerged Hydraulic Jump over Triangular Macroroughnesses." *Water (Switzerland)* 13(5).

[19] Henderson, F.M., 1966, *Open channel flow*, *Macmillan, New York*.

[20] Tokyay, Nuray Denli. 2005. "Effect of Channel Bed Corrugations on Hydraulic Jumps." *World Water Congress 2005: Impacts of Global Climate Change - Proceedings of the 2005 World Water and Environmental Resources Congress*: 408.

[21] Subramanya, K. 2009. "Flow In Open Channels. 3rd Edition", *Tata Mc Grow-Hill, New Delhi*, 344-351.

[22] Anderson, J.D., 1995. *Solutions Manual to Accompany Computational Fluid Dynamics: The Basics with Applications*. *McGraw-Hill*.

[23] Versteeg, H.K., 2007. *An introduction to computational fluid dynamics the finite volume method*, 2/E. *Pearson Education India*.

- [24] Canonsburg, T D. 2013. "ANSYS Fluent Theory Guide." *ANSYS Inc., USA* 15317(November):814.http://www.afs.enea.it/project/neptunius/docs/fluent/html/th/main_pre.htm.
- [25] Wilcox, D.C., 2006. One-equation and two-equation Models. Turbulence modeling for CFD, 1, pp.192-209.
- [26] France, P. W. 1981. "Analysis of the Hydraulic Jump Within a Diverging Rectangular Channel." *Proceedings of the Institution of Civil Engineers (London). Part 1 - Design & Construction* 71(pt 2): 369–78.
- [27] Hughes, William C., and J. Ernest Flack. 1984. "Hydraulic Jump Properties Over a Rough Bed." *Journal of Hydraulic Engineering* 110(12): 1755–71
- [28] Hager, Willi H., and Nicola V. Bretz. 1986. "Ressauts Hydrauliques Par-Dessus Des Marches Positives et Négatives." *Journal of Hydraulic Research* 24(4): 237–53.
- [29] Hager, Willi H., and Robert Wanoschek. 1987. "Ressaut Hydraulique En Canal Triangulaire." *Journal of Hydraulic Research* 25(5): 549–64.
- [30] Long, Dejiang, Nallamuthu Rajaratnam, Peter M. Steffler, and Peter R. Smy. 1991. "Structure de l'écoulement Dans Les Ressauts Hydrauliques." *Journal of Hydraulic Research* 29(2): 207–18.
- [31] Rajaratnam, N. 1995. 22 Canadian Journal of Civil Engineering *Energy Dissipators and Hydraulic Jump*.
- [32] Alhamid, Abdulaziz A, and Abdelazim M Negm. 1998. "Prediction of Sequent Depth Ratio of Hydraulic Jump in Abruptly Expanding Channel." (January).
- [33] Wu, S. and Rajaratnam, N., 1996. "Transition from hydraulic jump to open channel flow." *Journal of Hydraulic Engineering*, 122(9), pp.526-528.
- [34] Ead, S. A., and N. Rajaratnam. 2002. "Hydraulic Jumps on Corrugated Beds." *Journal of Hydraulic Engineering* 128(7): 656–63.

- [35] Omid, M. H., M. Esmaeeli Varaki, and Ranga Narayanan. 2007. "Gradually Expanding Hydraulic Jump in a Trapezoidal Channel." *Journal of Hydraulic Research* 45(4): 512–18.
- [36] Carollo, Francesco Giuseppe, Vito Ferro, and Vincenzo Pampalone. 2007. "Hydraulic Jumps on Rough Beds." *Journal of Hydraulic Engineering* 133(9): 989–99.
- [37] Chanson, Hubert, and Carlo Gualtieri. 2008. "Similitude and Scale Effects of Air Entrainment in Hydraulic Jumps." *Journal of Hydraulic Research* 46(1): 35–44.
- [38] Abbaspour, A., D. Farsadizadeh, A. H. Hosseiniadeh Dalir, and A. A. Sadraddini. 2009. "Numerical Study of Hydraulic Jumps on Corrugated Beds Using Turbulence Models." *Turkish Journal of Engineering and Environmental Sciences* 33(1): 61–72.
- [39] Abbaspour, A., A. Hosseinzadeh Dalir, D. Farsadizadeh, and A. A. Sadraddini. 2009. "Effect of Sinusoidal Corrugated Bed on Hydraulic Jump Characteristics." *Journal of Hydro-Environment Research* 3(2): 109–17.
- [40] Debabeche, Mahmoud, Sonia Cherhabil, Amin Hafnaoui, and Bachir Achour. 2009. "Hydraulic Jump in a Sloped Triangular Channel." *Canadian Journal of Civil Engineering* 36(4): 655–58.
- [41] Elsebaie, Ibrahim H, and Shazy Shabayek. 2010. "Formation of Hydraulic Jumps on Corrugated Beds." (01): 40–51.
- [42] Chachereau, Yann, and Hubert Chanson. 2011. "Free-Surface Fluctuations and Turbulence in Hydraulic Jumps." *Experimental Thermal and Fluid Science* 35(6): 896–909. <http://dx.doi.org/10.1016/j.expthermflusci.2011.01.009>.
- [43] Valiani, Alessandro, and Valerio Caleffi. 2011. "Linear and Angular Momentum Conservation in Hydraulic Jump in Diverging Channels." *Advances in Water Resources* 34(2): 227–42. <http://dx.doi.org/10.1016/j.advwatres.2010.11.006>.

- [44] Tokyay, Nuray Denli, T. U. Evcimen, and C. Şimşek. 2011. “Forced Hydraulic Jump on Non-Protruding Rough Beds.” *Canadian Journal of Civil Engineering* 38(10): 1136–44.
- [45] Afzal, Noor, A. Bushra, and Abu Seena. 2011. “Analysis of Turbulent Hydraulic Jump over a Transitional Rough Bed of a Rectangular Channel: Universal Relations.” *Journal of Engineering Mechanics* 137(12): 835–45.
- [46] Chanson, Hubert. 2012. “Momentum Considerations in Hydraulic Jumps and Bores.” *Journal of Irrigation and Drainage Engineering* 138(4): 382–85.
- [47] Tokyay, N D, and D Velioğlu. 2012. “Effects of Bed Roughness on the Characteristics of Hydraulic Jumps.” (October): 17–19.
- [48] Samadi-Boroujeni, H., M. Ghazali, B. Gorbani, and R. Fattahi Nafchi. 2013. “Effect of Triangular Corrugated Beds on the Hydraulic Jump Characteristics.” *Canadian Journal of Civil Engineering* 40(9): 841–47.
- [49] Abbaspour, Akram, Davood Farsadizadeh, and Mohammad Ali Ghorbani. 2013. “Estimation of Hydraulic Jump on Corrugated Bed Using Artificial Neural Networks and Genetic Programming.” *Water Science and Engineering* 6(2): 189–98.
- [50] Xiang, M., S. C.P. Cheung, J. Y. Tu, and W. H. Zhang. 2014. “A Multi-Fluid Modelling Approach for the Air Entrainment and Internal Bubbly Flow Region in Hydraulic Jumps.” *Ocean Engineering* 91: 51–63.
- [51] Ahmed, Hossam Mohamed Ali et al. 2014. “Effect of Corrugated Beds on Characteristics of Submerged Hydraulic Jump.” *Ain Shams Engineering Journal* 5(4): 1033–42. <http://dx.doi.org/10.1016/j.asej.2014.06.006>.
- [52] Esmaeili Varaki, M., A. Kasi, J. Farhoudi, and D. Sen. 2014. “Hydraulic Jump in a Diverging Channel with an Adverse Slope.” *Iranian Journal of Science and Technology - Transactions of Civil Engineering* 38(C1): 111–21.
- [53] Chanson, Hubert, and John Z. Shi. 2015. “Environmental Hydraulic Engineering: A Perspective.” *Environmental Fluid Mechanics* 15(3): 627–31.

- [54] Kateb, Samir, Mahmoud Debabeche, and Ferhat Riguet. 2015. "Hydraulic Jump in a Sloped Trapezoidal Channel." *Energy Procedia* 74: 251–57. <http://dx.doi.org/10.1016/j.egypro.2015.07.591>.
- [55] Bayon-Barrachina, Arnau, and Petra Amparo Lopez-Jimenez. 2015. "Numerical Analysis of Hydraulic Jumps Using OpenFOAM." *Journal of Hydroinformatics* 17(4): 662–78.
- [56] Kim, Youngkyu, Gyewoon Choi, Hyoseon Park, and Seongjoon Byeon. 2015. "Hydraulic Jump and Energy Dissipation with Sluice Gate." *Water (Switzerland)* 7(9): 5115–33.
- [57] Luo, Ching-ruey. 2015. "Hydraulic Jump----the State of the Art Hydrodynamic Characteristics of Fixed and Movable Beds." *International Journal of Applied Engineering and Technology* 1(1) https://www.researchgate.net/publication/279533608_International_Journal_of_Applied_Engineering_and_Technology.
- [58] Wang, Hang, and Hubert Chanson. 2016. "Self-Similarity and Scale Effects in Physical Modelling of Hydraulic Jump Roller Dynamics, Air Entrainment and Turbulent Scales." *Environmental Fluid Mechanics* 16(6): 1087–1110.
- [59] Mahtab, Ghorban, and Mohammad Taghi Sattari. 2016. "Investigation of Hydraulic Jump Characteristics in Rough Beds Using M5 Model Tree" *Jordan Journal of Agricultural Sciences* 12(2): 631–48.
- [60] Hafnaoui, M A, R F Carvalho, and M Debabeche. 2016. "Prediction of Hydraulic Jump Location in Some Types of Prismatic Channels Using Numerical Prediction of Hydraulic Jump Location in Some Types of Prismatic Channels Using Numerical Modelling." (June).
- [61] Sauda, Mohamed F. 2016. "Prediction of Hydraulic Jump Length Downstream of Multi-Vent Regulators Using Artificial Neural Networks." *Ain Shams Engineering Journal* 7(2): 819–26. <http://dx.doi.org/10.1016/j.asej.2015.12.005>.

- [62] Abbaspour, Akram, Sahar Parvini, and Ali Hosseinzadeh Dalir. 2016. "Effect of Buried Plates on Scour Profiles Downstream of Hydraulic Jump in Open Channels with Horizontal and Reverse Bed Slopes." *Water Science and Engineering* 9(4): 329–35. <http://dx.doi.org/10.1016/j.wse.2017.01.003>.
- [63] Hassanpour, Nasrin, Ali Hosseinzadeh Dalir, Davod Farsadizadeh, and Carlo Gualtieri. 2017. "An Experimental Study of Hydraulic Jump in a Gradually Expanding Rectangular Stilling Basin with Roughened Bed." *Water (Switzerland)* 9(12): 1–23.
- [64] Bejestan, Mahmood Shafai, and Manoochehr Shokrian Hajibehzad. 2018. "Predicting the Sequent Depth Ratio of a B-F Hydraulic Jump on a River-Bed Rock Chute." *Proceedings of the Institution of Civil Engineers: Water Management* 171(3): 143–51.
- [65] Valero, Daniel, Nicolò Viti, and Carlo Gualtieri. 2018. "Numerical Simulation of Hydraulic Jumps. Part 1: Experimental Data for Modelling Performance Assessment." *Water (Switzerland)* 11(1): 1–16.
- [66] Eshkou, Z., A. A. Dehghani, and A. Ahmadi. 2018. "Forced Hydraulic Jump in a Diverging Stilling Basin Using Angled Baffle Blocks." *Journal of Irrigation and Drainage Engineering* 144(8): 06018004.
- [67] Felder, Stefan, and Hubert Chanson. 2018. "Air–Water Flow Patterns of Hydraulic Jumps on Uniform Beds Macroroughness." *Journal of Hydraulic Engineering* 144(3): 04017068.
- [68] Salmasi, Farzin, and Aylar Samadi. 2018. "Experimental and Numerical Simulation of Flow over Stepped Spillways." *Applied Water Science* 8(8): 1–11. <https://doi.org/10.1007/s13201-018-0877-5>.
- [69] Jesudhas, Vimaldoss, Ram Balachandar, Vesselina Roussinova, and Ron Barron. 2018. "Turbulence Characteristics of Classical Hydraulic Jump Using DES." *Journal of Hydraulic Engineering* 144(6): 04018022.
- [70] Al-Hashimi, Shaymaa Abdul Muttaleb, Khitam Abdulhussein Saeed, and Thameen N. Nahi. 2019. "Experimental and CFD Modeling of Hydraulic Jumps

Forming at Submerged Weir.” *Journal of The Institution of Engineers (India): Series A* 100(3): 487–93. <https://doi.org/10.1007/s40030-019-00381-z>.

[71] Kumar, Mohit, Sanjay Kumar, and Sahil Bidhu. 2019. “Determination of Sequent Depth of Hydraulic Jump over Sloping Floor with Rounded and Crushed Aggregates Using Experimental and ANN Model.” *Water Science and Technology: Water Supply* 19(8): 2240–47.

[72] Torkamanzad, Nafiseh, Ali Hosseinzadeh Dalir, Farzin Salmasi, and Akram Abbaspour. 2019. “Hydraulic Jump below Abrupt Asymmetric Expanding Stilling Basin on Rough Bed.” *Water (Switzerland)* 11(9): 1–29.

[73] Yildiz, Ali, Ali Ihsan Marti, Alpaslan Yazar, and Volkan Yilmaz. 2020. “Determination of Position of Hydraulic Jump in a Flume by Using CFD and Comparison with Experiential Results <https://doi.org/10.21698/Rjeec.2020.211> P.” *Romanian Journal of Ecology & Environmental Chemistry* 2(2): 78–85.

[74] Jesudhas, Vimaldoss, Ram Balachandar, and Tirupati Boliseti. 2020. “Numerical Study of a Symmetric Submerged Spatial Hydraulic Jump.” *Journal of Hydraulic Research* 58(2): 335–49.

[75] Macián-Pérez, Juan Francisco, Arnau Bayón, et al. 2020. “Characterization of Structural Properties in High Reynolds Hydraulic Jump Based on CFD and Physical Modeling Approaches.” *Journal of Hydraulic Engineering* 146(12): 04020079.

[76] Ghaderi, Amir, Mehdi Dasineh, Francesco Aristodemo, and Ali Ghahramanzadeh. 2020. “Characteristics of Free and Submerged Hydraulic Jumps over Different Macroroughnesses.” *Journal of Hydroinformatics* 22(6): 1554–72.

[77] Hien, Le Thi Thu, and Duong Hoai Duc. 2020. “Numerical Simulation of Free Surface Flow on Spillways and Channel Chutes with Wall and Step Abutments by Coupling Turbulence and Air Entrainment Models.” *Water (Switzerland)* 12(11).

[78] Mahtabi, Ghorban, Barkha Chaplot, Hazi Mohammad Azamathulla, and

Mahesh Pal. 2020. "Classification of Hydraulic Jump in Rough Beds." *Water (Switzerland)* 12(8): 1–11.

[79] Nandi, B., S. Das, and A. Mazumdar. 2020. "Experimental Analysis and Numerical Simulation of Hydraulic Jump." *IOP Conference Series: Earth and Environmental Science* 505(1).

[80] Macián-Pérez, Juan Francisco, Francisco José Vallés-Morán, et al. 2020. "Experimental Characterization of the Hydraulic Jump Profile and Velocity Distribution in a Stilling Basin Physical Model." *Water (Switzerland)* 12(6).

[81] Nikmehr, Saman, and Younes Aminpour. 2020. "Numerical Simulation of Hydraulic Jump over Rough Beds." *Periodica Polytechnica Civil Engineering* 64(2): 396–407.

[82] Wiest, Rafael André et al. 2020. "Start Position of a Sloping Hydraulic Jump." *Revista Brasileira de Recursos Hidricos* 25: 1–9.

[83] Hafnaoui, Mohammed Amin, and Mahmoud Debabeche. 2021. "Numerical Modeling of the Hydraulic Jump Location Using 2D Iber Software." *Modeling Earth Systems and Environment* 7(3): 1939–46. <https://doi.org/10.1007/s40808-020-00942-3>.

[84] Gualtieri, C., and H. Chanson. 2021. "Physical and Numerical Modelling of Air-Water Flows: An Introductory Overview." *Environmental Modelling and Software* 143(June): 105109. <https://doi.org/10.1016/j.envsoft.2021.105109>.

[85] Maleki, Shayan, and Virgilio Fiorotto. 2021. "Hydraulic Jump Stilling Basin Design over Rough Beds." *Journal of Hydraulic Engineering* 147(1): 04020087.

[86] Nasrabadi, Mohsen, Yaser Mehri, Amin Ghassemi, and Mohammad Hossein Omid. 2021. "Predicting Submerged Hydraulic Jump Characteristics Using Machine Learning Methods." *Water Supply* 21(8): 4180–94.

[87] Türker, Umut, and Manousos Valyrakis. 2021. "Hydraulic Jump on Rough Beds: Conceptual Modeling and Experimental Validation." *Water Science and Technology: Water Supply* 21(4): 1423–37.

- [88] Hassanpour, Nasrin, Ali Hosseinzadeh Dalir, Arnau Bayon, and Milad Abdollahpour. 2021. "Pressure Fluctuations in the Spatial Hydraulic Jump in Stilling Basins with Different Expansion Ratio." *Water (Switzerland)* 13(1): 1–15.
- [89] Dasineh, M., Ghaderi, A., Bagherzadeh, M., Ahmadi, M., & Kuriqi, A. 2021. "Prediction of hydraulic jumps on a triangular bed roughness using numerical modeling and soft computing methods." *Mathematics*, 9(23), 3135
- [90] Wüthrich, Davide, Rui Shi, and Hubert Chanson. 2022. "Hydraulic Jumps with Low Inflow Froude Numbers: Air–Water Surface Patterns and Transverse Distributions of Two-Phase Flow Properties." *Environmental Fluid Mechanics* 22(4): 789–818. <https://doi.org/10.1007/s10652-022-09854-5>.
- [91] Parsamehr, Parastoo et al. 2022. "Hydraulic Jump over an Adverse Slope Controlled by Different Roughness Elements." *Water Resources Management* 36(14): 5729–49. <https://doi.org/10.1007/s11269-022-03330-x>.
- [92] Bahmanpouri, Farhad, Carlo Gualtieri, and Hubert Chanson. 2023. "Experiments on Two-Phase Flow in Hydraulic Jump on Pebbled Rough Bed: Part 1–Turbulence Properties and Particle Chord Time and Length." *Water Science and Engineering* (xxx): 1–11. <https://doi.org/10.1016/j.wse.2023.05.002>.
- [93] Zaffar, Muhammad Waqas, and Ishtiaq Hassan. 2023. "Numerical Investigation of Hydraulic Jump for Different Stilling Basins Using FLOW-3D." *Aqua Water Infrastructure, Ecosystems and Society* 72(7): 1320–43.
- [94] Macián-Pérez, Juan Francisco, Rafael García-Bartual, P. Amparo López-Jiménez, and Francisco José Vallés-Morán. 2023. "Numerical Modeling of Hydraulic Jumps at Negative Steps to Improve Energy Dissipation in Stilling Basins." *Applied Water Science* 13(10): 1–15. <https://doi.org/10.1007/s13201-023-01985-4>.
- [95] Navier, C.L., 1838. Navier Stokes Equation. Paris: Chez Carilian-Goeury.
- [96] Celik, I. B., Ghia, U., Roache, P. J., & Freitas, C. J. (2008). Procedure for estimation and reporting of uncertainty due to discretization in CFD applications.

Journal of fluids Engineering-Transactions of the ASME, 130(7).

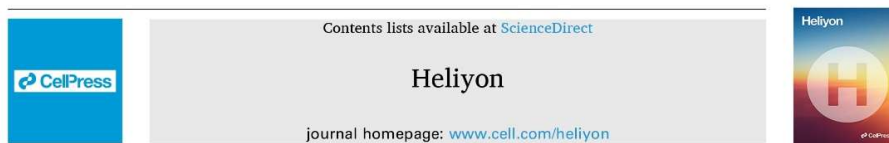
LIST OF PUBLICATIONS

Journals SCI/SCIE

1. Harshit Kumar Jayant, Bharat Jhamnani, Numerical simulation of free and submerged hydraulic jump over trapezoidal and triangular macroroughness, Heliyon, Volume 9, Issue 11, 2023, e22540, ISSN 2405-8440, <https://doi.org/10.1016/j.heliyon.2023.e22540> (SCI-Expanded)
2. Harshit Kumar Jayant, Bharat Jhamnani; Numerical study of submerged hydraulic jumps over triangular macroroughnesses. Journal of Hydroinformatics 1 January 2024; 26 (1): 51–71. <https://doi.org/10.2166/hydro.2023.302>. (SCI-Expanded)

International Conferences

1. CFD Investigation of Hydraulic Jump over Triangular Roughness. Recent advances in Sustainable Construction technologies and Environment-2022. B.S. Abdur Rahman Crescent Institute of Science and Technology.
2. Effect of mesh and turbulence models on predicting the real-world fluid phenomenon using numerical simulation. International Conference on Next Generations Digital Technologies, Akal University, Talwandi Sabo, Bathinda, Punjab.



Numerical simulation of free and submerged hydraulic jump over trapezoidal and triangular macroroughness

Harshit Kumar Jayant^{*}, Bharat Jhamnani

Delhi Technological University, Department of Civil Engineering, Delhi 110042, India

ARTICLE INFO

Keywords:
Computational fluid dynamics
Hydraulic jump
Macroroughness

ABSTRACT

Hydraulic jump is a phenomenon that occurs in open channel flow and has wide utility in hydraulic engineering. The present study considered macroroughness features in the bed to improve the characteristics of free and submerged hydraulic jumps. For that, the most often utilized triangular macroroughness elements are contrasted with a distinctive trapezoidal macroroughness. This study implied a numerical model based on computational fluid dynamics (CFD) to investigate the hydraulic jump characteristics over the macroroughness and represented the important features of hydraulic jump such as sequent depth ratio, tail-water depth ratio, longitudinal velocity profile, flow pattern of velocity vector, and contours for turbulent kinetic energy over smooth and macrorough beds. It is seen that for a specific Froude number, triangular macroroughness outperforms trapezoidal macroroughness. The average value of the length of the roller for trapezoidal and triangular macroroughness is found to be 23 % and 46.06 % shorter than the smooth bed in the case of a free jump, whereas 17.85 % and 36.33 % shorter than the smooth bed in the case of a submerged jump, respectively. To check the accuracy of the numerical model, the basic parameters of a hydraulic jump, such as sequent depth and tail-water depth ratio, are compared with the experimental findings of the present work, and the mean error between the CFD and experimental models is reported to be within 6 %, which confirms the reliability of the CFD model and can be applied to real hydraulic engineering.

1. Introduction

A hydraulic jump is a phenomenon that occurs in open-channel flow, typically involving the flow of water in rivers, channels, or spillways. This phenomenon is characterized by a rapid decrease in flow velocity and an abrupt increase in water depth over a relatively short distance. Classical hydraulic jumps are the most common type and are associated with abrupt energy dissipation and the formation of turbulent rollers. Classical hydraulic jumps were studied in depth by Refs. [1–6]. Further hydraulic jumps are classified into two categories free and submerged hydraulic jumps. A submerged hydraulic jump takes place below the water's surface, while a free hydraulic jump happens above it [7]. Fig. 1 depicts the definition sketch of hydraulic jump and their important parameters where, a sluice gate is provided at the boundary of the upstream and downstream sections in order to illustrate the free and submerged hydraulic jump phenomenon. The sluice gate functions as a hydraulic structure to release extra water from the reservoir, which transforms potential energy into kinetic energy, upstream portion contains reservoir conditions. To reduce the risk of damaging to the downstream bed and hydraulic system, this extra energy must be treated close to it. The hydraulic jump is a common occurrence that

^{*} Corresponding author.

E-mail address: harshit_2k19phdce04@dtu.ac.in (H.K. Jayant).

<https://doi.org/10.1016/j.heliyon.2023.e22540>

Received 31 August 2023; Received in revised form 14 November 2023; Accepted 14 November 2023

Available online 21 November 2023

2405-8440/© 2023 The Authors. Published by Elsevier Ltd. This is an open access article under the CC BY-NC-ND license (<http://creativecommons.org/licenses/by-nc-nd/4.0/>).

[illegible]

Numerical study of submerged hydraulic jumps over triangular macroroughnesses

Harshit Kumar Jayant * and Bharat Jhamnani

Department of Civil Engineering, Delhi Technological University, Delhi, India

*Corresponding author. E-mail: harshit_2k19phdce04@dtu.ac.in

 HKJ, 0009-0002-3493-2925

ABSTRACT

The hydraulic jump is a phenomenon that occurs in open channels. In past studies, hydraulic jumps over smooth and macrorough beds have been investigated to enhance energy dissipation, but triangular macroroughness, specifically the right-angled triangular macroroughness, has not been dealt with. The objective of this article is to numerically investigate submerged hydraulic jumps over right angle and isosceles triangular macrorough beds. To achieve this, a numerical model based on computational fluid dynamics (CFD) has been utilized. Numerically obtained jump characteristics such as submerged depth ratio, tailwater depth ratio, longitudinal velocity profile, flow pattern in the cavity region, and energy dissipation have been presented in detail. In particular, initial Froude number reduction in both tailwater and submerged depth ratios as well as an increase in the energy dissipation of submerged hydraulic jumps have been noticed on isosceles triangular macroroughness with different arrangements, as compared to smooth beds. The present numerical model has been validated with the experimental model, and the mean error between the two for submerged depth and tailwater depth ratios was found to be below 6%. This confirms the adequacy of the present CFD model in predicting relevant submerged hydraulic jump characteristics over macrorough beds.

Key words: ANSYS fluent, computational fluid dynamics, submerged hydraulic jump, triangular macroroughness

HIGHLIGHTS

- Isosceles triangular shapes are more commonly used, and right-angled shapes have not been addressed in the past.
- Right-angled triangular macroroughness is added as a new shape of triangular configuration.
- Different macroroughness arrangements are tested to enhance the jump characteristics, which have not been extensively explored in the past.

This is an Open Access article distributed under the terms of the Creative Commons Attribution Licence (CC BY-NC-ND 4.0), which permits copying and redistribution for non-commercial purposes with no derivatives, provided the original work is properly cited (<http://creativecommons.org/licenses/by-nc-nd/4.0/>).

**SCHOOL OF INFRASTRUCTURE
DEPARTMENT OF CIVIL ENGINEERING**

Certificate of Appreciation

This is to certify that **Mr./Ms. HARSHIT KUMAR JAYANT** from **DELHI TECHNOLOGICAL UNIVERSITY DELHI** has presented a paper on “**CFD INVESTIGATION OF HYDRAULIC JUMP OVER TRIANGULAR ROUGHNESS**” at the Two Day Virtual International Conference on “**Recent Advances in Sustainable Construction Technologies and Environment, (ICRASCE-2022)**” organized by Department of Civil Engineering, School of Infrastructure, B.S. Abdur Rahman Crescent Institute of Science and Technology on 15th & 16th of June 2022.



Co-ordinators

**Dr. V S Priya/ Dr. Nisha Khanam/
Mr Dhamodharan S**



Conveners

**Dr. M. S. Haji Sheik Mohammed
Dr Vasanthi P**



Registrar

Dr. N. Raja Hussain



Talwandi Sabo



International Conference on Next Generation Digital Technologies
Akal University, Talwandi Sabo, Bathinda
(January 3-5, 2024)

CERTIFICATE OF PRESENTATION

THIS CERTIFICATE IS PROUDLY PRESENTED TO
HARSHIT KUMAR JAYANT

of

Delhi Technological University

For Successfully Presented a Paper on

"Effect of Mesh and Turbulence Models on Predicting the Real World Fluid Flow Phenomenon Using Numerical Simulation"

In the "International Conference on Next Generation Digital Technologies", organized by Department of Computer Science & Engineering, Akal University, Talwandi Sabo, Bathinda, Punjab in Collaboration with Eternal University, Baru Sahib and Central University of Punjab Bathinda.

Dr. Upinder Kaur
Convener
ICNGDT-2024



Dr. Namrop Kaur
Co-Convener
ICNGDT-2024

PLAGIARISM REPORT



Page 1 of 131 - Cover Page

Submission ID trn:oid::27535:99818124

Harshit Kumar Jayant**Harshit Kumar Jayant PhD Thesis.pdf**

Delhi Technological University

Document Details

Submission ID

trn:oid::27535:99818124

Submission Date

Jun 8, 2025, 9:54 AM GMT+5:30

Download Date

Jun 8, 2025, 10:04 AM GMT+5:30

File Name

Harshit Kumar Jayant PhD Thesis.pdf

File Size

7.6 MB

121 Pages

25,987 Words

140,037 Characters



Page 1 of 131 - Cover Page

Submission ID trn:oid::27535:99818124

6% Overall Similarity

The combined total of all matches, including overlapping sources, for each database.





Filtered from the Report

- Bibliography
- Small Matches (less than 10 words)




Exclusions

- 3 Excluded Sources
- 9 Excluded Matches

Match Groups

-  **117 Not Cited or Quoted 5%**
Matches with neither in-text citation nor quotation marks
-  **12 Missing Quotations 1%**
Matches that are still very similar to source material
-  **0 Missing Citation 0%**
Matches that have quotation marks, but no in-text citation
-  **0 Cited and Quoted 0%**
Matches with in-text citation present, but no quotation marks

Top Sources

- 2%  Internet sources
- 3%  Publications
- 4%  Submitted works (Student Papers)

Integrity Flags

0 Integrity Flags for Review

No suspicious text manipulations found.

Our system's algorithms look deeply at a document for any inconsistencies that would set it apart from a normal submission. If we notice something strange, we flag it for you to review.

A Flag is not necessarily an indicator of a problem. However, we'd recommend you focus your attention there for further review.

BRIEF CURRICULUM VITAE

Harshit Kumar Jayant is currently working as a Junior Design Engineer (Civil) at the Research Designs and Standards Organisation (RDSO), Lucknow, under the Ministry of Railways, Government of India. He completed his Bachelor of Technology in Civil Engineering from the Kalinga Institute of Industrial Technology, Bhubaneswar in 2016, followed by a Master of Technology in Hydraulics and Water Resources Engineering from Delhi Technological University (DTU), Delhi in 2018. In 2019, he joined the Doctor of Philosophy programme in Civil Engineering at DTU, Delhi.

His academic and research interests are cantered on hydraulic engineering, with a focus on tackling emerging challenges using Computational Fluid Dynamics (CFD) modelling. Additionally, he has contributed extensively to Railway engineering, addressing issues related to track safety and track monitoring systems on the Indian Railways. Throughout his career, he has emphasized practical engineering problem-solving through analytical, computational, and applied research approaches.