MODELING FLUID FLOW IN OPEN DRAIN CHANNEL USING COMPUTATIONAL FLUID DYNAMICS (CFD)

By

OSEMEAHON ESEOHE ABRAHAM

ENG2002134

DEPARTMENT OF CIVIL ENGINEERING,

FACULTY OF ENGINEERING,

UNIVERSITY OF BENIN,

EDO STATE

UNIVERSITY OF BENIN,

BENIN CITY

NOVEMBER, 2025

PLAGIARISM

This work, MODELING FLUID FLOW IN OPEN DRAIN CHANNELS USING CFD by OSEMEAHON, Eseohe Abraham with mat number ENG2002134 of the Department of Civil Engineering, University of Benin, Edo State, Nigeria, has PASSED the PLAGIARISM TEST.

PROJECT COORDINATOR:

NAME: …………………………………………………………………………………………….

SIGNATURE AND DATE: ……………………………………………………………………………

CERTIFICATION

This is to certify that this project work titled "**Modelling Fluid flow in Open drain channel using CFD**" was carried out by **OSEMEAHON ESEOHE ABRAHAM** with Matriculation number, **ENG2002134**, of the Department of Civil Engineering, Faculty of Engineering, University of Benin, Benin-City, Edo State, Nigeria.

……………………………………. **………………………….**

ENGR. ORIAKHI OROBOSA **DATE**

(SUPERVISOR)

……………………………………. **……………………………**

ENGR. PROF. NGOZI IHIMEKPEN **DATE**

(HEAD OF DEPARTMENT)

DEDICATION

I dedicate this project to God Almighty for giving me the strength and wisdom throughout my academic pursuit. Special dedication to my parents, Mr. and Mrs. Osemeahon, whose love, care and understanding have been a constant motivation. I also extend the dedication to my siblings, friends, and mentors for their encouragement.

ACKNOWLEDGEMENT

All glory be to God Almighty, whose grace and strength have carried me through this academic journey. I sincerely thank my supervisor, Engr. Dr. Oriakhi Orobosa, for his unending guidance and encouragement throughout the course of this research. I also wish to acknowledge all the staffs of the department of Civil Engineering, their dedication to excellence and their continuous encouragement which has created an enabling environment for learning. I also wish to express my acknowledgement to my project coordinator; Engr. E Oria-Usifo and all my lecturers; Engr. Prof. O.U. Orie, Engr. Prof. O.C. Izinyon, Engr. Prof. H.A.P. Audu, Engr. Prof. J.O. Okovido, Engr. Prof. S.D. yeke, Engr. Prof R.I. Umasabor, Engr. Dr. R.O. Ogirigbo, Engr. Dr. (Mrs) Kayode-Ojo, Engr. Dr. P.N. Ogbiefun, Engr. Dr. (Mrs) L.O. Bobor, Engr. Dr. A. Agbonaye, Engr. Dr. R. Ilaboya, Engr. Dr. U. Ukeme, Engr. Dr. (Mrs) A. Rawlings, Engr. B. Omosefe, Engr. C. Okolie, Engr. O. Osasu, Engr. N. Oghoyafedo, Engr. (Mrs) E. Ambrose-Agabi, Engr. E. Musa, Engr. Dr. I. Iziengbe, Engr. U. Ogbonna, Engr. (Mrs) GE Evbaru, Engr. J.O. Odemerho and the entire non-academic staff of the Department of Civil Engineering, University of Benin, for their contributions, whether through knowledge impartation, encouragement, and the academic environment that facilitated this research.

I would also like to thank the academic and non-academic staff of the Department of Civil Engineering, University of Benin, for their dedication and contributions to my learning.

Finally, I appreciate my family and friends for all the love, support and encouragement throughout the course of this work.

ABSTRACT

Open drain channels play a vital role in stormwater management and flood prevention in urban and agricultural environments. However, factors such as complex channel geometry, surface roughness, sediment accumulation, and turbulence often reduce their efficiency, leading to flooding and waterlogging. This study aimed to analyze fluid flow in an open drainage channel using Computational Fluid Dynamics (CFD) to understand flow behavior, identify hydraulic inefficiencies, and recommend design improvements for enhanced drainage performance.

The study focused on an open drainage channel within the University of Benin, Ugbowo Campus, Benin City, Nigeria. Data on channel dimensions, flow conditions, and physical characteristics were obtained through site investigations and measurements. A three-dimensional model of the channel was developed and simulated using CFD techniques in SolidWorks Flow Simulation. The governing equations of fluid flow, including the continuity and Navier-Stokes equations, were solved under appropriate boundary conditions to evaluate velocity distribution and free-surface flow characteristics.

The simulation results showed that flow velocity increased gradually along the channel length, with the highest velocities occurring near the outlet region. The free surface remained stable throughout the simulation, indicating efficient water conveyance under the specified flow conditions. The study concluded that CFD is an effective tool for predicting flow behavior and evaluating the hydraulic performance of open drainage systems. The findings provide valuable insights for improving drainage design, reducing flood risks, and promoting sustainable stormwater management.

<u>**TABLE OF CONTENT**</u>

CHAPTER TWO

LITERATURE REVIEW

LIST OF TABLES

LIST OF FIGURES

LIST OF NOMECLATURE

CFD - Computational Fluid Dynamics

RANS - Reynolds-Averaged Navier-Stokes

LES - Large Eddy Simulation

SUDS - Sustainable Urban Drainage Systems

PIV - Particle Image Velocimetry

VOF - Volume of Fluid

DNS - Direct Numerical Simulation

FVM - Finite Volume Methods

DES - Detached Eddy Simulation

GES - Geographic Information System

CHAPTER ONE

INTRODUCTION

1.1 Background of Study

The analysis of fluid flow in open drain channels has become an essential aspect of modern hydraulic engineering and infrastructure management. Open drain channels serve a crucial role in managing surface runoff, directing stormwater, and preventing issues such as flooding and waterlogging. These systems are particularly relevant in urban and agricultural settings, where efficient water drainage is critical for safeguarding property, reducing health risks associated with stagnant water, and supporting sustainable land use practice (Cho, Dao, & Nichols, 2022). However, fluid flow in open drain channels is influenced by a variety of complex factors, including the geometry of the channel, surface roughness, and flow turbulence. These factors are dynamic and interdependent, making it challenging to predict flow behavior using traditional analytical methods. Historically, physical modeling and empirical equations have been employed to analyze fluid dynamics in open channels. While these approaches have proven useful in certain contexts, they often fail to capture the full complexity of real-world conditions (Naves, Anta, Suárez, & Puertas, 2020). For instance, physical models can be costly, time-consuming, and difficult to scale, while empirical methods frequently rely on simplifying assumptions that may not accurately reflect actual flow scenarios. To address these limitations, Computational Fluid Dynamics (CFD) has emerged as a transformative tool for simulating and analyzing fluid flow. CFD involves the numerical solution of the governing equations of fluid mechanics, specifically the Navier-Stokes equations. These equations describe the motion of fluid particles and account for variables such as velocity, pressure, and turbulence. By utilizing advanced algorithms and computational power, CFD allows for the detailed visualization and analysis of fluid flow in complex systems, including open drain channels (Shafique & Kim, 2017). One of the primary advantages of CFD is its ability to simulate a wide range of flow conditions with high accuracy. Unlike physical models, CFD eliminates the need for costly prototypes and enables researchers to study multiple design alternatives under controlled conditions. Additionally, CFD provides detailed insights into flow patterns, pressure distributions, and energy dissipation, which are critical for optimizing drainage system performance. For example, CFD can identify regions of flow separation or recirculation within a channel, which may contribute to inefficiencies and potential blockages (Chanson, 2015). Despite its advantages, the application of CFD in hydraulic engineering is not without challenges. The accuracy of CFD simulations depends on several factors, including the quality of the computational mesh, the selection of appropriate turbulence models, and the definition of boundary conditions. Moreover, the computational cost of running high-resolution simulations can be significant, particularly for large-scale or transient flow scenarios. However, advances in computational hardware and software have made CFD increasingly accessible, enabling its widespread adoption in both academic research and practical engineering applications (Ferziger and Perić, 2020).

This study aims to leverage CFD to analyze fluid flow in an open drain channel, focusing on the identification of inefficiencies, the prediction of hydraulic performance, and the recommendation of design improvements. By applying CFD techniques, the project seeks to bridge the gap between theoretical knowledge and practical application, contributing to the development of more efficient and sustainable drainage systems. Ultimately, the findings of this research have the potential to support better water resource management practices, enhance infrastructure resilience, and promote sustainable urban and agricultural development (Guo, 2021 and Chanson, 2015).

1.2 Statement of problem

The management of fluid flow in open drain channels poses significant challenges in modern hydraulic and environmental engineering. These channels, which play a critical role in preventing flooding, directing stormwater, and maintaining surface runoff, often fail to perform optimally due to the complexities of fluid dynamics. Inefficient flow management can lead to sediment deposition, flow separation, recirculation zones, and energy losses, all of which compromise the effectiveness of drainage systems and increase the risks of urban flooding and waterlogging (Chanson, 2015). These problems are exacerbated in regions prone to intense rainfall, where poorly designed channels are unable to handle peak discharge rates effectively (Zafarmomen, Samadi & Borgomeo, 2026).

Traditional approaches to analyzing flow behavior in open drain channels such as empirical formulae, physical modeling, and field measurements offer limited predictive accuracy and often require substantial time and financial investment. Physical models, while valuable for visualizing fluid flow, are difficult to scale and may not capture the intricate turbulence effects that occur in real-world scenarios. Additionally, empirical equations typically rely on simplifications and assumptions that neglect the spatial and temporal variations in fluid properties and boundary conditions. These limitations hinder engineers' ability to optimize drainage channel designs effectively. Furthermore, the complexity of fluid interactions, such as turbulence and transient flow phenomena, complicates the analysis of hydraulic performance in open channels.

Turbulence, for instance, can result in energy dissipation and uneven velocity distribution, making it difficult to ensure efficient discharge and minimize erosion or sedimentation (Li & Mo, 2026). Transient flow phenomena, including rapid changes in flow rate or velocity during storm events, introduce additional challenges for accurate modeling and prediction (Ferziger and Perić, 2020). Addressing these issues requires advanced tools capable of capturing both steady-state and dynamic flow behaviors with high precision. Computational Fluid Dynamics (CFD) provides a promising solution to these challenges by enabling detailed simulation of fluid flow in complex geometries. Unlike traditional methods, CFD can model three-dimensional flow patterns, predict hydraulic performance under various conditions, and evaluate design alternatives efficiently. However, despite its potential, the application of CFD to open drain channels remains underutilized, particularly in developing regions where resource constraints limit access to advanced computational tools (Yang, Cao, Dai, Xia, & Xu, 2026). Additionally, the accuracy of CFD simulations often depends on proper election of turbulence models, boundary conditions, and computational grids, highlighting the need for rigorous validation and calibration (Chanson, 2015).

This study seeks to address these gaps by applying CFD to model fluid flow in open drain channels, focusing on understanding flow behavior, identifying inefficiencies, and recommending design improvements. By leveraging CFD's capabilities, the project aims to provide actionable insights into optimizing drainage systems for enhanced performance and sustainability. The results are expected to contribute to improved water resource management practices and infrastructure resilience, particularly in areas vulnerable to flooding and climate change impacts.

1.3 Aims and Objectives

The primary aim of this study is to develop an accurate computational model to simulate fluid flow dynamics in open drain channels using Computational Fluid Dynamics (CFD) techniques.

The specific objectives are to;

(i) Develop and validate CFD models to simulate fluid flow under various conditions, such as flow rates.

(ii) Investigate the impacts of design parameters and obstructions on fluid flow patterns and drainage efficiency.

(iii) Compare simulated results with experimental or field data to ensure the accuracy and reliability of the models.

1.4 Scope of Study

(i) Channel configurations : Study diverse open drain channel geometrics like rectangular, trapezoidal and irregular shapes.

(ii) Simulation and Validation: Model fluid flow under varying conditions and validate CFD results using experimental or field data.

(iii) Practical Applications: Address real-world challenges like flood control, drainage efficiency, and erosion mitigation.

(iv) Design Optimization: Explore channel design improvements to enhance performance and tackle issues such as stagnation and overflow.

1.5 Justification of Study

The study of fluid flow in open drain channels using Computational Fluid Dynamics (CFD) is of paramount importance for addressing various engineering, environmental, and urban challenges. Urbanization has drastically increased impervious surfaces, leading to higher stormwater runoff and placing immense pressure on drainage systems. Inefficient or poorly designed open drain channels contribute to severe flooding, causing loss of life, infrastructural damage, and socio-economic disruptions. Through this study, critical tools are provided to analyze and refine channel designs, enabling more efficient water conveyance, particularly during high flow conditions.

Traditional methods of drainage analysis often rely on costly physical models and extensive field studies, making them resource intensive and time consuming. CFD modeling offers a transformative alternative by enabling precise simulations of fluid flow under diverse scenarios, facilitating the optimization of channel design and performance without the need for physical testing. This cost effective approach supports sustainable engineering practices, ensuring optimal use of resources while minimizing environmental impacts. The environmental implications of poor drainage systems include soil erosion, sediment deposition, water stagnation, and harm to local ecosystems (Yu & Kitsikoudis, 2026). By providing detailed insights into fluid flow dynamics, this study aids in mitigating such impacts, offering solutions to preserve ecosystems, control sediment transport, and enhance agricultural productivity. With accurate CFD models, engineers can design channels that address these challenges effectively. The study also equips policymakers and engineers with reliable data to inform critical decisions in urban planning, infrastructure design, and emergency flood response. Predictive capabilities gained through CFD simulations allow communities to build resilience against flooding and other water related crises, protecting both infrastructure and livelihoods. Moreover, the increasing intensity and unpredictability of rainfall due to climate change demand adaptive and resilient drainage systems. This research contributes to climate adaptation strategies by analyzing the performance of open drain channels under extreme conditions and supporting future-proof designs. (Chen, Wang & Xu 2019).

Furthermore, the integration of CFD in hydraulic engineering promotes innovation by advancing modeling techniques and expanding their applicability. By bridging the gap between theoretical research and practical engineering challenges, this study lays a solid foundation for efficient, resilient, and sustainable drainage system designs, benefiting communities, ecosystems, and professionals in the field. Its outcomes are anticipated to have lasting impacts on urban infrastructure, water management practices, and environmental conservation efforts.

CHAPTER TWO

LITERATURE REVIEW

2.1 Overview of Open Drain Flow Dynamics

Open drain channels, prevalent in urban and rural drainage systems, exhibit complex flow dynamics influenced by various factors such as channel geometry, surface roughness, and flow conditions. Understanding these dynamics is crucial for effective design, operation, and maintenance of drainage infrastructure.

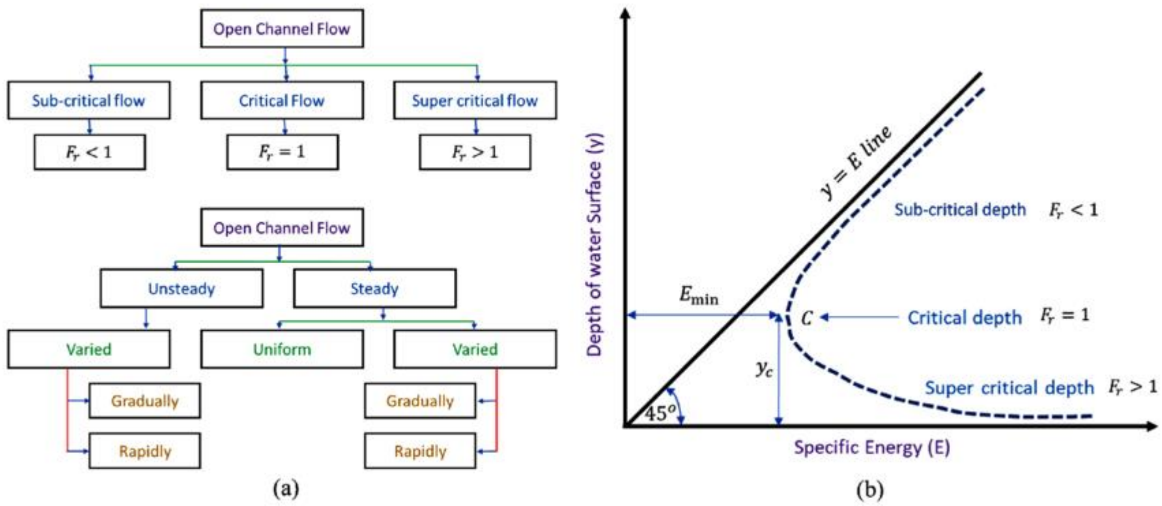
2.1.1 Fundamental Characteristics of Open Channel Flow

Open channel flow is characterized by the presence of a free surface exposed to atmospheric pressure, distinguishing it from pressurized pipe flow. The flow is primarily driven by gravity, with the flow depth and velocity varying along the channel length. The flow can be classified based on temporal and spatial variations:

Steady vs. Unsteady Flow: Steady flow implies that flow parameters (depth, velocity) remain constant over time at a given location, whereas unsteady flow indicates temporal variations.

Uniform vs. Non-uniform Flow: Uniform flow occurs when flow parameters are constant along the channel length, while non-uniform flow involves spatial variations.

These classifications are essential for selecting appropriate analytical or numerical methods for flow analysis (Proust, Berni & Nikora, 2025).



Types of open channel flow **Specific Energy curve**

FIG 2.1 Overview of open drain flow dynamics

2.1.2 Empirical Formulas for Velocity Estimation

Empirical formulas have been developed to estimate flow velocity in open channels, accounting for factors like channel slope, roughness, and hydraulic radius:

Manning's Equation: Introduced by Robert Manning in 1890, this formula estimates average flow velocity based on channel slope, roughness coefficient, and hydraulic radius. It's widely used due to its simplicity and applicability to various channel shapes (Shounda, & Debnath, 2026).

Chézy's Formula: Developed by Antoine Chézy in the 18th century, this formula relates flow velocity to the square root of the product of hydraulic radius and channel slope, incorporating a coefficient determined experimentally (Naves, Anta, & Puertas, 2020).

These formulas provide foundational tools for preliminary design and analysis of open channel flows.

2.1.3 Turbulence and Shear Stress

Turbulence plays a significant role in open channel flow dynamics, influencing momentum transfer, mixing, and energy dissipation. Shear stress, resulting from velocity gradients near the channel bed and walls, affects sediment transport and channel stability. Rouse (1937) highlighted the importance of turbulence in sediment suspension and transport processes.

Tang and Knight (2008) developed analytical solutions using depth-integrated Navier-Stokes equations to evaluate shear stress distribution in compound channels, considering bed friction, lateral turbulence, and secondary currents. Their work emphasizes the complexity of flow structures in natural channels with varying geometries.

2.1.4 Unsteady Flow Phenomena and Hydraulic Structures

Unsteady flows, such as those caused by storm events or sudden releases from reservoirs, can lead to rapid changes in flow depth and velocity. Hydraulic jumps, a common unsteady flow phenomenon, occur when supercritical flow transitions to subcritical flow, resulting in a sudden rise in water surface elevation and energy dissipation through turbulence (Unal, 2018).

Understanding these phenomena is vital for designing hydraulic structures like spillways, energy dissipators, and flood control channels to ensure safety and functionality during variable flow conditions.

2.1.5 CFD in Open Channel Flow Analysis

Advancements in computational methods have enabled detailed analysis of open channel flows using CFD. CFD allows for the simulation of complex flow patterns, including turbulence, secondary currents, and interactions with structures. Mohammadian et al. (2020) reviewed various CFD studies on effluent discharges, highlighting the application of Reynolds-Averaged Navier-Stokes (RANS) and Large Eddy Simulation (LES) models in predicting flow behavior.

These models facilitate the investigation of flow characteristics in scenarios where analytical solutions are challenging, such as irregular geometries and transient flow conditions.

2.1.6 Challenges in modeling

Modeling open drain systems presents several intricate challenges due to the complex interplay of hydrodynamic and sediment transport processes. Key issues include accurately representing turbulence, sediment dynamics, and variable boundary conditions, all of which are critical for reliable simulations.

(i) Sediment Transport Dynamics

Sediment transport in open drains involves complex mechanisms such as entrainment, suspension, and deposition, influenced by flow velocity, sediment size, and turbulence intensity. Modeling these processes requires integrating sediment transport equations with hydrodynamic models, considering factors like settling velocity and bed shear stress. Accurate representation of sediment dynamics is essential, as sediment deposition can reduce channel capacity and affect flood risk (Gaudio, 2020).

(ii) Variable Boundary Conditions

Open drain systems often experience unsteady flow conditions due to rainfall events, inflow variations, and downstream obstructions. These variable boundary conditions complicate modeling efforts, as they require time-dependent simulations and adaptive boundary treatments. Accurately capturing these dynamics is crucial for predicting flow behavior and designing effective drainage infrastructure (Alihosseini, & Thamsen, 2019).

(iii) Model Validation and Data Limitations

Validating models for open drain systems is challenging due to limited availability of high-resolution field data. Discrepancies between model predictions and observed data can arise from simplifications in model assumptions and uncertainties in input parameters. Continuous monitoring and data collection are essential to improve model reliability and inform calibration efforts (Mohammadian et al., 2020).

Addressing these challenges requires a multidisciplinary approach, combining advanced modeling techniques with comprehensive field studies to enhance the understanding and management of open drain systems.

2.2 Traditional Methods of Flow Analysis

Traditional methods of flow analysis in open channels have been foundational in hydraulic engineering, providing empirical and analytical tools for predicting flow behavior. These methods, developed over centuries, remain integral to the design and analysis of open channel systems.

2.2.1 Empirical approaches

Empirical methods are grounded in observational data and are often favored for their simplicity and ease of application. Among the most widely used empirical formulas are the Chézy and Manning equations

(i) Chezy's Formula

Antoine Chezy introduced one of the earliest empirical formulas for open channel flow in the 18th century. The Chezy equation relates the mean flow velocity (V) to the channel's hydraulic radius (R) and slope (S) through a coefficient (C):

$$V = C \sqrt{(R\ S)}$$

The Chezy coefficient (C) encapsulates the effects of channel roughness and flow resistance. While pioneering, the formula's reliance on an empirical coefficient limited its adaptability across diverse channel conditions. Subsequent refinements led to more versatile models, yet Chezy's work laid the groundwork for future developments in open channel hydraulics .

(ii) Manning's Equation

Building upon Chezy's foundation, Robert Manning developed an equation in 1889 that introduced a more practical approach to estimating flow velocity in open channels. The Manning equation is expressed as:

$V = (1/n) R^{(2/3)} S^{(1/2)}$

Here, 'n' represents the Manning roughness coefficient, a parameter accounting for the channel's surface roughness. This equation gained widespread acceptance due to its simplicity and applicability to various channel types, including natural streams and engineered canals. Its empirical nature, however, necessitates careful selection of the 'n' value to ensure accuracy.

While empirical methods are invaluable for quick assessments and preliminary designs, their accuracy is contingent upon the appropriate selection of coefficients, which can vary based on channel conditions and flow regimes.

2.2.2 Analytical Approaches

Analytical methods derive from fundamental principles of fluid mechanics, offering a more rigorous framework for understanding open channel flows. These approaches are particularly beneficial when dealing with complex flow scenarios or when high precision is required.

(i) Darcy-Weisbach Equation

The Darcy-Weisbach equation, initially formulated for pipe flow, has been adapted for open channel applications. It calculates head loss due to friction as:

$hf = f (L/D) (V^2/2g)$

Where 'hf' is the head loss, 'f' is the Darcy friction factor, 'L' is the length of the channel, 'D' is the hydraulic diameter, 'V' is the flow velocity, and 'g' is the acceleration due to gravity. This equation offers a more theoretically grounded approach compared to empirical formulas, but its application in open channels requires accurate determination of the friction factor, which can be complex due to varying flow conditions and channel geometries.

(ii) Saint-Venant Equations

For unsteady and non-uniform flow conditions, the Saint-Venant equations provide a comprehensive framework. These equations consist of the continuity and momentum equations, accounting for temporal and spatial variations in flow depth and velocity. They are particularly useful in modeling flood waves, dam breaks, and other transient flow phenomena. However, their complexity often necessitates numerical methods for solutions, such as finite difference or finite element techniques.

In summary, traditional methods of open channel flow analysis, ranging from empirical formulas like Chezy's and Manning's to more analytical approaches like the Darcy-Weisbach and Saint-Venant equations, have been instrumental in advancing hydraulic engineering. Each method offers unique advantages and limitations, and their applicability depends on the specific characteristics of the channel and flow conditions. A thorough understanding of these methods enables engineers to select the most appropriate tools for designing and analyzing open channel systems.

2.2.3 Limitations of manual or simplified methods

Manual and simplified methods have long been utilized in open channel flow analysis due to their straightforward application and minimal computational requirements. However, these approaches come with several limitations that can impact the accuracy and reliability of hydraulic designs. (Stum, 2010).

(i) Oversimplification of Complex Flow Conditions

Simplified methods often assume uniform flow conditions and steady-state scenarios, which may not accurately represent real-world situations. For instance, the Manning equation presumes uniform flow and does not account for rapidly varying flows or complex channel geometries. This can lead to significant errors in predicting water surface profiles and flow velocities, especially in natural channels with irregular cross-sections.

(ii) Limited Applicability to Irregular Channel Geometries

Manual methods typically rely on idealized channel shapes, such as rectangular or trapezoidal sections. When applied to natural channels with irregular geometries, these methods may not capture the true hydraulic behavior. This limitation can result in inaccurate estimations of flow parameters, which are critical for flood risk assessments and infrastructure design.

(iii) Inadequate Representation of Unsteady Flow Conditions

Manual calculations are generally not equipped to handle unsteady flow conditions, such as those caused by storm events or dam breaches. The inability to model transient flow behaviors can hinder the accurate prediction of flood waves and the design of effective mitigation measures.

(iv) Time-Consuming and Prone to Human Error

Manual computations, especially for complex systems, can be time-consuming and susceptible to human error. The iterative nature of some calculations increases the likelihood of mistakes, which can compromise the integrity of the analysis and subsequent designs.

(v) Challenges in Modeling Sediment Transport and Erosion

Simplified methods often do not account for sediment transport dynamics and channel erosion processes. Neglecting these factors can lead to underestimations of sediment deposition or scour, potentially resulting in the failure of hydraulic structures or unintended environmental impacts.

In conclusion, while manual and simplified methods offer ease of use, their limitations in handling complex, unsteady, and irregular flow conditions necessitate the adoption of more advanced modeling techniques. Incorporating computational tools and numerical models can enhance the accuracy and reliability of open channel flow analyses, leading to better-informed engineering decisions. (Mohammadian and Garcia, 2020).

2.3 CFD in Open Channel Flow Modeling

CFD is a branch of fluid mechanics that uses numerical analysis and algorithms to solve and analyze problems involving fluid flows. By discretizing the governing equations of fluid motion—namely the Navier-Stokes equations—CFD allows simulation of fluid dynamics under various boundary and initial conditions (Wang, Cao, Huang, et al. 2024). The Navier-Stokes equations describe the conservation of mass, momentum, and energy in a fluid. When applied to open channel flows, these equations must be solved with appropriate assumptions and simplifications, especially when modeling shallow water conditions where the flow depth is significantly smaller than the horizontal dimensions.

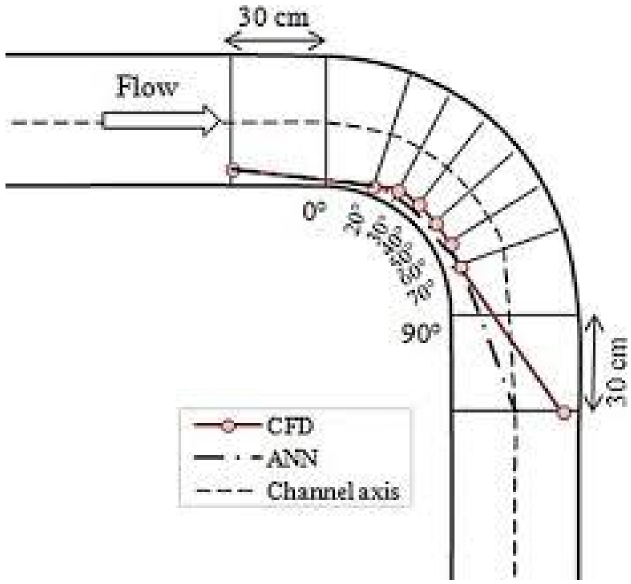


Fig 2.2 Simulation of Open channel bend characteristics using Computational Fluid Dynamics

2.3.1 Application in Open Channel Flow

CFD in open channel flow modeling is used to analyze flow patterns, velocity distributions, sediment transport, pollutant dispersion, and interactions with hydraulic structures. The primary motivations include:

(i) Predicting water levels and velocities in natural streams and engineered channels.

(ii) Assessing flood risks and designing flood control structures.

(iii) Evaluating the impact of hydraulic structures such as weirs, culverts, and spillways (Mkhonta, Byaruhanga & Kibirige, 2026).

(iv) Models range from one-dimensional (1D) to three-dimensional (3D), with 3D models providing the most detailed insight but requiring greater computational resources.

2.3.2 Benefits of CFD over Traditional Methods

The development and application of Computational Fluid Dynamics (CFD) have revolutionized the field of fluid mechanics and hydraulic engineering. Compared to traditional methods which include analytical solutions, empirical formulas, and physical modeling, CFD offers significant advantages in terms of precision, flexibility, cost-effectiveness, and visualization. These benefits have led to its widespread adoption in industries ranging from water resources and civil engineering to aerospace and biomedical sciences.

(a) Enhanced Accuracy and Detail

Traditional methods, such as analytical models or empirical equations (e.g., Manning's formula or Darcy-Weisbach equation), are based on simplifying assumptions like uniform flow, steady-state conditions, or idealized geometries (Chow, 1959). These simplifications can limit their applicability in complex or transient flow situations. CFD, on the other hand, solves the Navier-Stokes equations numerically and can capture the full three-dimensional, time-dependent behavior of fluid flows. This allows engineers to analyze localized flow features such as turbulence, vortices, and flow separation around structures, which are often neglected in traditional methods (Versteeg & Malalasekera, 2007). According to Ferziger and Perić (2002), CFD can provide insights into flow phenomena that are either impossible or too expensive to capture experimentally. This includes internal flow dynamics in hydraulic machines, pollutant dispersion in urban drainage, and the detailed flow around bridge piers and spillways.

(b) Ability to Handle Complex Geometries and Boundary Conditions

Traditional hydraulic models often struggle to represent irregular geometries, such as natural riverbeds, urban floodplains, or engineered structures with complicated shapes. Physical models can be constructed but are typically scaled-down and expensive to fabricate. CFD allows precise representation of complex geometries through mesh generation and boundary condition customization. Using tools like finite volume or finite element solvers, engineers can simulate flow around obstructions, through porous media, or over rough terrain (Patankar, 1980). Lai (2000) demonstrated the application of unstructured CFD models in riverine environments, highlighting how flexible mesh configurations allow accurate simulation of natural channels, which is unfeasible with rigid grid-based traditional methods.

(c) Cost and Time Efficiency

While CFD simulations can be computationally expensive, they are often more cost-effective and quicker than constructing physical models or conducting extensive field experiments, especially when multiple scenarios or design iterations are required. Munson et al. (2013) noted that in design optimization problems, CFD allows rapid prototyping and testing of different configurations virtually, thereby saving resources in the early stages of project development. Moreover, CFD can simulate extreme or hazardous conditions such as dam-break flows, toxic gas dispersion, or high-velocity jet impingement without risk to human life or infrastructure (Wang et al., 2012).

(d) Flexibility in Multiphysics Coupling

Another major advantage of CFD is its ability to couple multiple physical processes within a single model. For example, CFD can integrate:

(i) Sediment transport and bed morphodynamics (Wu, 2004)

(ii) Heat transfer in thermal pollution studies (Incropera et al., 2007)

(iii)Chemical reactions in pollutant modeling

(iv) Structural interactions in fluid-structure interaction (FSI) analyses (Bazilevs et al., 2006)

These multiphysics capabilities are difficult or impossible to replicate with traditional methods or simplified numerical models.

(e) Scalability and Scenario Testing

CFD enables scenario testing under different boundary conditions, structural designs, and climate change impacts. For example, flood modeling using traditional 1D methods (e.g., HEC-RAS) may suffice for simple channels but cannot simulate overland flow or urban stormwater flooding effectively.

Neal et al. (2012) showed that 2D and 3D CFD-based flood models can simulate subgrid-scale features and infrastructure interactions more reliably. This scalability makes CFD especially useful in urban planning, risk assessment, and real-time emergency response simulations.

These developments are steadily bridging the gap between CFD and traditional tools in terms of accessibility and ease of use.

2.4 Review of Key CFD Studies in Open Drain Flow

Open drain systems channels designed to carry stormwater, wastewater, or surface runoff play a crucial role in urban and rural drainage infrastructure. Accurately modeling the flow behavior in open drains is vital for assessing hydraulic performance, flood risks, pollutant transport, and sediment deposition. Traditional modeling methods, such as empirical or 1D analytical approaches, often fall short in capturing the full spatial variability of flow patterns. Over the past two decades, Computational Fluid Dynamics (CFD) has emerged as a powerful tool for analyzing and optimizing flow in open drain systems. This review highlights key CFD-based studies that have advanced the understanding and simulation of open drain flows, particularly in urban hydrology, environmental hydraulics, and sediment transport modeling.

(i) CFD Modeling of Stormwater Flow in Urban Open Drains

One of the earlier comprehensive applications of CFD in open drain flow was by Lai (2000), who developed a three-dimensional finite volume model for free-surface flows in natural and engineered channels. The study demonstrated how unstructured grids could simulate irregular open drain geometries and complex inlet/outlet conditions—features that are difficult to handle in traditional methods. In a study of urban drainage systems, Gironás et al. (2009) compared 1D hydrologic models (like SWMM) with CFD simulations for open drains under varying rainfall events. Their research showed that CFD provided more detailed insight into velocity profiles, hydraulic jumps, and backwater effects at junctions—critical for infrastructure design and retrofitting.

(ii) Hydraulic Performance and Design Optimization

Tominaga and Nezu (1991) applied Reynolds-Averaged Navier-Stokes (RANS) models to open channel flows, including open drains, to study turbulence structures and secondary currents. Their findings helped improve the understanding of how channel roughness and shape influence flow resistance important for designing efficient drains with minimal clogging. In the context of stormwater management, Marques et al. (2010) used CFD to analyze how various drain configurations affected flow velocities and shear stresses. Their study emphasized how improper design of open drains could lead to erosion at channel bends or deposition in low-velocity zones.

(iii) CFD in Sediment Transport and Erosion Studies

Sediment transport modeling is a critical concern in open drain maintenance. Wu (2004) developed a depth-averaged 2D CFD model that integrated sediment continuity equations with hydrodynamics to simulate non-uniform sediment transport. This model has since been applied to analyze siltation in urban open drains and propose sediment flushing strategies. In another study, Khosronejad et al. (2011) used a high-resolution 3D CFD model to study local scour near culverts and drain outfalls. Their research showed how flow turbulence and eddy formation contributed to sediment detachment, helping inform erosion control measures.

(iv) Vegetation Effects and Flow Resistance in Drains

Vegetation often plays a dual role in open drain flow: it can reduce flow velocity and enhance sedimentation, but also increase resistance and cause clogging. Nepf (2012) provided a CFD-based framework for modeling vegetation–flow interactions using drag-based formulations in the RANS framework. Later, Defina and Bixio (2005) applied a two-layer model in CFD to simulate flow through vegetated channels, representing stem density and height as resistance parameters. Their work is especially relevant for bioengineered open drains in Sustainable Urban Drainage Systems (SUDS).

(v) Integration of CFD with Field and Experimental Data

CFD models have been increasingly validated against laboratory and field data to improve reliability. For example, Kang and Sotiropoulos (2011) compared CFD simulations of open channel and drain flows with experimental data from Particle Image

Velocimetry (PIV), showing close agreement in velocity distribution and turbulence kinetic energy. Validation efforts like these have enhanced the credibility of CFD models in practical applications such as urban flood control, drain retrofitting, and environmental impact assessments.

(vi) Recent Innovations: Multiphase Flow and Real-Time Modeling

Recent CFD studies have expanded into modeling multiphase flows (e.g., water–air or water–debris) in open drains. Yang et al. (2018) developed a Volume of Fluid (VOF) model to simulate drain overflow under heavy rainfall conditions, capturing dynamic air–water interface movement. Moreover, researchers like Ghaheri et al. (2021) are exploring real-time CFD modeling by integrating machine learning with traditional solvers, aiming to predict flow and pollution transport in smart urban drainage networks.

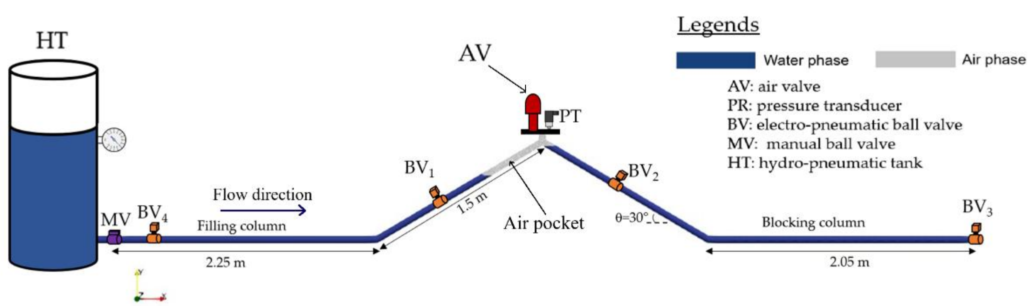


Fig 2.3 Digital twin based on CFD modelling for analysis of Two-phase flows during pipeline fittings

2.4.1 Modeling Techniques Used in CFD

Computational Fluid Dynamics (CFD) offers a suite of modeling techniques tailored to simulate the complex behaviors of flow in open channels and drainage systems. The choice between 2D and 3D modeling approaches, as well as turbulence modeling strategies such as the Reynolds-Averaged Navier-Stokes (RANS) equations, depends on the goals of the study, geometric complexity, and available computational resources. This section explores the principal modeling techniques, with academic references to key contributions in the field.

(a) 2D vs. 3D Modeling Approaches

2D Modeling Techniques

Two-dimensional (2D) CFD models simulate flow in a horizontal (x–y) plane while assuming uniform conditions in the vertical (z) direction. These models are often depth-averaged, simplifying the Navier-Stokes equations to reduce computational effort while maintaining acceptable accuracy in shallow, wide channels.

Wu (2004) developed a depth-averaged 2D model for sediment-laden flow in open channels. The model successfully captured flow velocity and bed evolution using a simplified approach, making it suitable for long-term morphological studies. 2D models are widely used in floodplain analysis and river hydraulics (Neal et al., 2012). They strike a balance between efficiency and resolution, making them ideal for large-scale applications or early-stage design. However, 2D models are limited in their ability to capture vertical structures such as secondary currents, stratification, or flow separation, which are critical in narrow, deep, or highly turbulent flows.

3D Modeling Techniques

Three-dimensional (3D) CFD models solve the full Navier-Stokes equations in all three spatial dimensions and are capable of resolving vertical velocity gradients and complex hydrodynamic structures.

According to Lai (2000), 3D models are essential for simulating localized flow behavior in open drains and around hydraulic structures like culverts and drop inlets. These models use structured or unstructured grids and are especially useful for analyzing mixing, scouring, or turbulent eddies.

Khosronejad et al. (2011) applied 3D models to simulate local scour around bridge piers and drain outlets, demonstrating the capability of 3D CFD in resolving detailed erosion patterns caused by flow turbulence.

Despite their accuracy, 3D models are computationally intensive and require finer mesh resolution, robust solvers, and significant post-processing, limiting their use to high-priority or small-scale studies.

(b) Turbulence Modeling: RANS, LES, and DNS

Flow in open channels and drains is inherently turbulent, especially during storms or rapid discharge events. To model turbulence, several techniques are used:

(i) Reynolds-Averaged Navier-Stokes (RANS) Models

RANS models are the most commonly used approach in hydraulic CFD. These models decompose instantaneous flow variables into mean and fluctuating components and solve time-averaged equations. Popular RANS models include:

k–ε model – Effective for simulating fully developed turbulent flow (Launder & Spalding, 1974).

k–ω model – More accurate near walls and separation zones (Wilcox, 2006).

RNG k–ε model – Incorporates effects of mean flow strain and has been used in complex open channel applications (Yakhot & Orszag, 1986).

Versteeg and Malalasekera (2007) highlighted the wide adoption of RANS models in environmental flows due to their computational efficiency and reasonable accuracy for a broad range of scenarios.

(ii) Large Eddy Simulation (LES)

LES resolves large energy-containing eddies explicitly while modeling smaller scales. It offers higher accuracy in simulating unsteady and anisotropic turbulence than RANS. Tominaga and Stathopoulos (2013) used LES in open channel simulations to study flow separation and reattachment zones, which are not well-captured by RANS models. While LES provides superior turbulence resolution, it requires finer meshes and smaller time steps, making it more suitable for research and validation studies than for routine engineering design.

(iii) Direct Numerical Simulation (DNS)

DNS resolves all turbulence scales without any modeling assumptions, solving the full Navier-Stokes equations directly. Though extremely accurate, DNS is computationally prohibitive and currently limited to fundamental research on turbulence in small-scale domains (Moin & Mahesh, 1998). It is rarely used in practical open drain or channel flow applications.

(c) Free Surface and Multiphase Flow Modeling

Many open channel flows involve free surface motion, such as waves, surges, and overflows. Common techniques to model this include:

(i) Volume of Fluid (VOF) method – Tracks the interface between air and water by solving an additional scalar transport equation (Hirt & Nichols, 1981).

(ii) Level Set method – Uses a signed distance function to capture the fluid interface, allowing better curvature resolution (Osher & Sethian, 1988).

(iii) Yang et al. (2018) applied VOF to simulate air–water interactions in stormwater drains under heavy rainfall. The model effectively captured surface splashing and overflow phenomena.

(d) Grid and Numerical Methods

CFD solvers use either structured, unstructured, or hybrid grids depending on the channel geometry. Finite Volume Methods (FVM) dominate in hydraulic applications due to their conservation properties and stability (Patankar, 1980). Moukalled et al. (2016) describe in detail how FVM is used to discretize the flow domain and numerically solve governing equations for open drain and river systems.

Table 2.1 Summary of technique suitability

Technique	**Application Suitability**	**Limitations**
2D Depth-Averaged	River/floodplain flows, large areas	Cannot resolve vertical structures
3D CFD	Local flow around structures, detailed turbulence	High computational demand
RANS	Engineering design, steady turbulent flow	Less accurate for complex unsteady flow
LES	Unsteady flow analysis,	Requires fine grid and small

	vortex studies	time step
VOF	Surface flows, overflow modeling	May diffuse sharp interface
DNS	Fundamental turbulence research	Not suitable for engineering scale problems

2.5 Identified Gaps and Research Opportunities

The application of Computational Fluid Dynamics (CFD) in modeling fluid flow in open drain channels has witnessed significant progress over the past decades. However, despite these advancements, several critical gaps remain that hinder comprehensive understanding and predictive accuracy in real-world scenarios.

(i) Limited Integration of Multiphase and Sediment Transport Models

One of the major limitations in current CFD modeling of open drain channels is the underrepresentation of multiphase flow and sediment transport processes. Most studies have focused on single-phase flow (e.g., water only), overlooking the complexities introduced by suspended sediments, debris, and pollutants (Guan et al., 2016; Fang et al., 2020). Sediment transport not only affects flow resistance and channel geometry over time but also impacts pollutant dispersion and ecological health, especially in urban drainage systems. Existing models often assume static or simplified boundary conditions, which do not reflect the dynamic nature of sediment-water interactions (Zhao et al., 2018).

(ii) Scalability and Real-Time Simulation Limitations

CFD simulations for open drain channels are computationally intensive, especially when accounting for 3D turbulence, transient flow conditions, and large-scale networks. While high-performance computing has mitigated some of these issues, scalability remains a concern. Real-time applications, such as flood prediction and drainage optimization, are still constrained by the long processing times required for complex models (Chen et al., 2019; Li and Zhang, 2021). There is an opportunity for research into hybrid modeling approaches combining CFD with machine learning or surrogate modeling—to reduce computational load without significantly sacrificing accuracy.

(iii) Insufficient Field Validation and Calibration

A persistent issue in CFD modeling is the scarcity of robust field data for model validation and calibration. Many studies rely on laboratory-scale experiments or hypothetical geometries, which may not accurately represent real-life drainage systems, particularly in regions with informal or aging infrastructure (Hassan et al., 2017). Field measurements are often hindered by logistical challenges and sensor limitations, leading to gaps in model accuracy and confidence (Singh and Bhallamudi, 2015). Future work should prioritize comprehensive data collection campaigns and explore the use of remote sensing and IoT technologies for real-time validation.

(iv) Turbulence Modeling Challenges

Accurate representation of turbulence in open drain channels, particularly under highly unsteady flow regimes (e.g., during storm events), remains a challenge. Although Reynolds-Averaged Navier-Stokes (RANS) models like k-ε and k-ω are widely used due to their relative simplicity, they often fail to capture flow separation, vortex shedding, and anisotropic turbulence, especially around channel junctions, bends, and sudden expansions (Patel et al., 2014; Lai and Khan, 2020). Large Eddy Simulation (LES) and Direct Numerical Simulation (DNS) offer more accuracy but at a higher computational cost. There is a need for novel turbulence models that strike a balance between fidelity and efficiency for practical engineering use.

(v) Neglect of Vegetative and Structural Interactions

Many open channels, particularly in semi-natural or urban environments, contain vegetation and infrastructural elements such as culverts, trash racks, and baffles. However, these features are frequently neglected or oversimplified in CFD models, potentially leading to significant errors in predicting flow behavior, energy loss, and pollutant trapping (Nepf, 2012; Moradi and Ghaemi, 2022). There exists a research gap in accurately modeling fluid-structure-vegetation interactions within CFD frameworks. This includes the need for dynamic modeling of flexible vegetation, which responds to flow forces and alters flow resistance accordingly.

(vi) Climate Change and Urbanization Impacts

The influence of climate change—such as increased rainfall intensity and frequency—combined with rapid urbanization has altered the hydraulic characteristics of open drainage systems. Yet, few CFD studies incorporate these evolving boundary conditions and their long-term impacts on drainage performance

(Shafique and Kim, 2017). There is a growing need to couple CFD models with urban growth and climate projection tools to assess the resilience of drainage systems under future scenarios.

(vii) User Accessibility and Model Usability

Finally, despite the technical advancements, the usability of CFD tools remains a concern for many practitioners. Steep learning curves, high costs of software, and the need for specialized expertise limit the widespread adoption of CFD in routine drainage design and management (Anderson et al., 2015). Future research should focus on developing user-friendly, open-source CFD platforms tailored for civil engineers and urban planners, with pre-set templates for common drainage configurations.

In summary, while CFD has revolutionized the modeling of fluid flow in open drain channels, several critical research opportunities persist. Future efforts should emphasize:

(i) Developing integrated multiphase models with sediment transport.

(ii) Enhancing scalability through hybrid and surrogate models.

(iii) Strengthening field data collection for calibration and validation.

(iv) Advancing turbulence models suitable for open channel conditions.

(v) Incorporating vegetation and infrastructure in high-resolution simulations.

(vi) Addressing long-term impacts of climate change and urban expansion.

(vii) Creating accessible, practitioner-oriented

2.6 Suggestions for Advancing CFD Use in Open Drain Modeling

Computational Fluid Dynamics (CFD) has proven to be a valuable tool in modeling flow behavior in open drainage systems, enabling detailed analysis of fluid motion, energy losses, sediment transport, and pollutant dispersion. However, its full potential in open drain applications has yet to be realized due to various limitations in model accuracy, computational resources, and data integration. The following suggestions aim to advance the application of CFD in this domain.

(i) Enhancing Turbulence Modeling for Shallow and Transitional Flows

CFD simulations of open channels and drains often struggle to accurately predict turbulence, especially in shallow or transitional flows where flow separation, eddies, and surface disturbances are significant. Traditional turbulence models like the k-ε or k-ω models have been widely used (e.g., Rodi, 1993), but they may not capture complex flow structures in drains with obstructions, junctions, or rapid transitions.

Suggestion: Incorporate more advanced turbulence models such as Large Eddy Simulation (LES) or Detached Eddy Simulation (DES), which can resolve larger turbulent structures and are more suited for unsteady and non-uniform flow in drains (e.g., Ferziger and Perić, 2002; Sagaut, 2006).

(ii) Coupling CFD with Sediment Transport and Pollutant Dispersion Models

While CFD can simulate hydrodynamics effectively, its integration with sediment transport and water quality models is often limited. Accurate simulation of sediment dynamics in urban drains is essential for predicting clogging, erosion, and contaminant spread.

Suggestion: Develop coupled models that integrate CFD solvers with sediment transport modules (e.g., Van Rijn, 1984) and pollutant dispersion algorithms. García (2008) emphasized the need for such coupling in sediment-laden flows to improve environmental assessments and infrastructure planning.

(iii) Improving Boundary Condition Specifications Using Real Time Data

One of the major challenges in CFD modeling of open drains is the accurate specification of inlet, outlet, and wall boundary conditions, particularly under unsteady flow conditions. Many studies assume steady or uniform conditions for simplicity (e.g., Gessner and Emery, 1981), which may not reflect reality.

Suggestion: Integrate CFD models with real-time monitoring data (e.g., from SCADA systems, IoT sensors) to dynamically update boundary conditions and enhance model accuracy (e.g., Liu et al., 2019).

(iv) Addressing Free Surface Tracking and Air-Water Interaction

Open drains involve a free water surface exposed to the atmosphere, and accurately tracking this interface is crucial, especially under unsteady or rapidly varied flow. Traditional CFD methods often use the Volume of Fluid (VOF) or Level Set methods, which can suffer from numerical diffusion or instabilities.

Suggestion: Employ advanced free surface capturing methods, such as improved VOF algorithms with geometric reconstruction (e.g., Hirt and Nichols, 1981), or hybrid approaches combining VOF and Level Set (e.g., Sussman et al., 1994). These methods enhance the precision of surface profile prediction, especially in surges and backflow zones.

(v) Validation with Experimental and Field Data

Model calibration and validation are essential for ensuring the reliability of CFD predictions. However, in many studies, validation is limited or based on laboratory conditions that may not reflect real drain behavior.

Suggestion: Encourage field-scale validation using actual measurements from open drainage networks. Kocabas and Aral (2002) stressed the importance of field validation in pollutant transport models, and similar emphasis should be placed in hydraulic modeling.

(vi) Integration with GIS and Urban Hydrology Models

Urban drainage systems are spatially distributed, and their performance is influenced by land use, topography, and rainfall patterns. Standalone CFD models often lack the spatial context needed for broader urban hydrological assessments.

Suggestion: Link CFD models with Geographic Information Systems (GIS) and hydrologic-hydraulic models like SWMM (Storm Water Management Model). Zoppou (2001) highlighted the need for integrated models in urban stormwater management, which can help simulate entire drainage networks with localized CFD analysis of critical points.

CHAPTER THREE

METHODOLOGY

Fig 3.1 Overview of the map of University of Benin campus, Ugbowo, Benin City, Nigeria

3.1 The study area

The study focuses on an open drainage channel within the University Of Benin (UNIBEN), Ugbowo Campus, in Benin City, Nigeria. The drain runs from the staff quarters, through residential areas and discharges near Ekosodin village. It experiences frequent flooding during heavy rainfall due to sediment buildup, poor maintenance, and rapid runoff from impervious surfaces.

The channel is rectangular, partially lined with concrete, and varies in width from 1.0 to 1.5 meters. During storm events, flow depths can reach up to 0.9 meters, with estimated peak discharges between 2.0 and 3.5 m^3/s. The study area is hydraulically complex, making it ideal for

CFD analysis to understand flow behavior, identify flood-prone zones, and propose design improvements for better stormwater management on campus.

3.1.1 Geographic Location

The selected study area is an open drainage channel located within the University Of Benin (UNIBEN), Ugbowo Campus, in Benin City, Edo State, Nigeria. This area was chosen due to its recurrent flooding issues during the rainy season, which affect student hostels, staff quarters, other residents and nearby roadways. The open drain in focus is part of the campus's stormwater conveyance network and receives runoff from a combination of paved and vegetated surfaces.

i. Coordinates:
   a) Latitude: 6.395°N – 6.404°N
   b) Longitude: 5.603°E – 5.615°E
ii. Total Drain Length: Approximately 800 meters

3.1.2 Physical and Structural Description of the Drain Channel

Fig 3.2 Overview of drain

The drainage channel under study runs from the Faculty of Engineering complex, passing through the Senior Staff Quarters, and discharges near the Ekosodin village boundary. The channel is partially lined with concrete and exhibits several transitions in cross-section and slope. Notable features include:

i. Channel Geometry: rectangular, with occasional box culverts
ii. Bottom Width: ~1.0 to 1.5 meters
iii. Side Slopes: 1V:2H
iv. Average Flow Depth:
    a) Dry Season: ~0.1 – 0.2 meters

b) Peak Rain Events: Up to 0.9 meters

v. Bed Slope: Ranges from 0.0015 to 0.004

vi. Surface Material: Mix of concrete, vegetation, and silt deposits

Obstructions such as waste buildup, overgrown vegetation, and partially collapsed sections are common, which hinder efficient flow and increase flooding risk.

3.2 Project Processes

This outline includes data collection, model setup, simulation, validation, and analysis, providing a clear roadmap for executing the project.

3.2.1 Site Investigation and Preliminary Assessment

i. Purpose: Understand the physical characteristics and hydraulic behavior of the drainage channel.

ii. Activities:

a) Conduct a reconnaissance survey of the drainage channel within UNIBEN, especially the segment from the Faculty of Engineering to Ekosodin boundary.

b) Identify critical sections with overflows, blockages, or erosion.

c) Take photographic and GPS records.

3.2.2 Data Collection

i. Topographic and Geometric Data

Collect elevation and cross-sectional profiles.

Map:

a) Channel width, depth, and slope
b) Presence of culverts, junctions, and flow obstructions
c) Channel lining type (e.g., concrete or unlined)

Measure or estimate:

a) Flow depths during wet and dry conditions
b) Flow velocities (manually or using a flow meter)
c) Manning's roughness coefficient for different sections

3.2.3 Drainage Channel Modeling

i. Geometry and Domain Creation

Develop a 3D model of the drainage channel using:

a) SolidWorks for initial geometry
b) Export geometry to mesh generation software

ii. Mesh Generation

Use tools like ICEM CFD or ANSYS Meshing to discretize the domain.

Choose mesh type:

a) Structured mesh for simple segments
b) Unstructured mesh for areas with complex geometry

Apply refined mesh near walls, bends, and culverts to capture flow features accurately.

3.2.4 CFD Model Setup

i. Software: Use ANSYS Fluent or OpenFOAM

Set up the governing equations:

a) Continuity equation

b) Navier-Stokes equations (incompressible flow)

ii. Choose flow model:

a) Laminar model for low flow

b) Turbulence model (e.g., k-ε or k-ω) for storm flow conditions

iii. Boundary Conditions

a) Inlet: Define based on estimated or measured inflow rate (time-dependent if transient)

b) Outlet: Set a pressure outlet or flow depth condition

c) Walls: Apply no-slip conditions

d) Include free surface modeling (e.g., Volume of Fluid – VOF method) for open channel flow

iv. Solver Settings

Define:

a) Time step and simulation duration (for transient cases)

b) Gravity effects

c) Convergence criteria (residuals $< 10^{-4}$ or 10^{-6})

3.2.5 Model Calibration and Validation

i. Compare simulation outputs (velocity, depth, flow profile) with:

a) Field measurements

b) Observed flood extent during past rainfall events

ii. Adjust:

a) Roughness values

b) Inlet hydrograph shape

c) Mesh resolution

3.2.6 Results Analysis

i. Visualize flow patterns:

  a) Velocity vectors and streamlines

  b) Water surface profiles

  c) Pressure distribution

  d) Turbulence zones and recirculation

ii. Identify:

a) Flood-prone areas (backwater effects, overflow points)

b) High-velocity regions that may cause erosion

c) Sediment deposition zones (if modeled)

3.2.7 Recommendations

Based on simulation results, propose improvements such as:

i. Channel redesign (e.g., wider cross-sections, smoother transitions)

ii. Installation of trash screens

iii. Reinforcement of eroded segments

iv. Periodic desilting schedule

v. Improved inlet/outlet structures

CHAPTER FOUR

RESULTS AND DISCUSSION

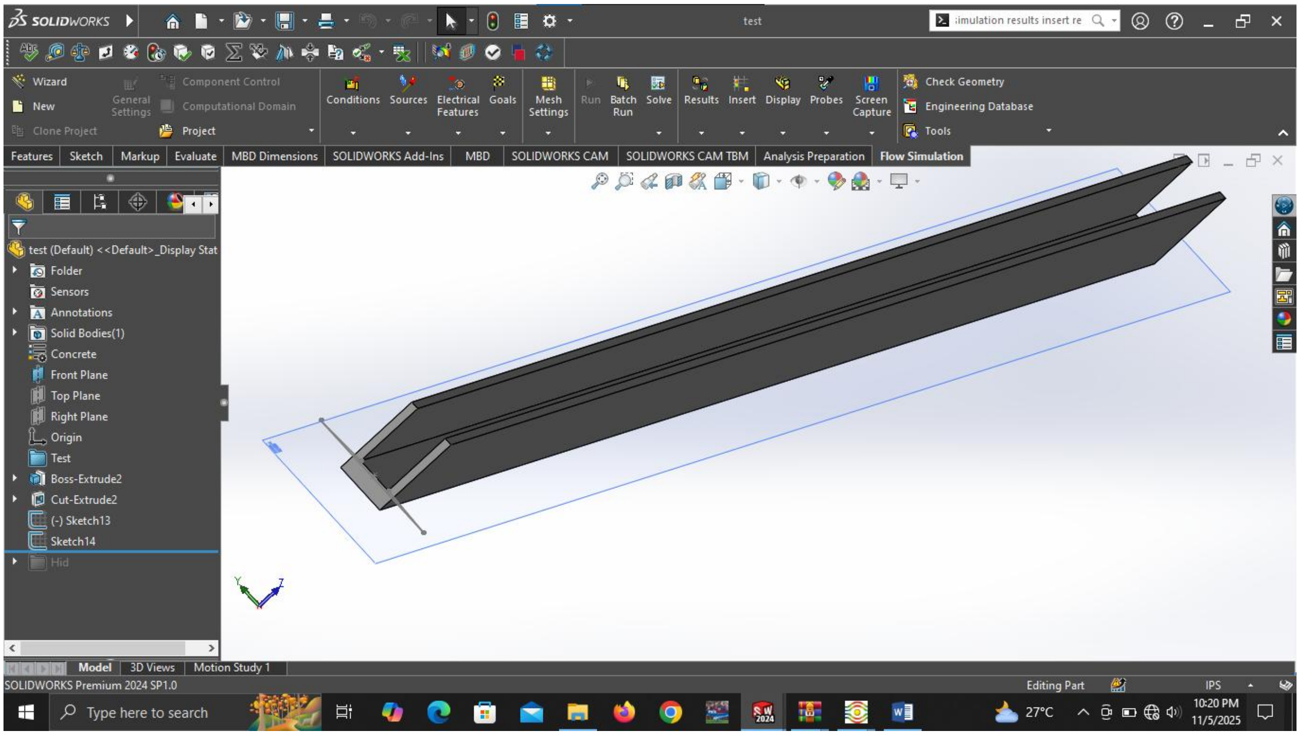


Fig 4.1 Model Setup and Simulation Overview

4.1 Drain Geometry and Flow Domain Description

The modeled drain is a straight open channel with a rectangular cross-section. The model represents a typical concrete drainage system used for stormwater conveyance. The dimensions used are:

Parameters	Value
Channel length	800m
Channel width	1.5m
Channel Depth	2.0m

4.2 Flow Simulation Project Conditions

The simulation was performed under realistic open-channel assumptions. It was set up as a two-phase free surface flow problem, consisting of water flowing through the channel while air occupies the region above the water surface.

The working fluids selected in the project were:

i. Water (liquid phase)
ii. Air (gas phase)

Gravity effects were enabled in the negative Y- direction to accurately represent open channel flow under natural conditions. $g=9.81m/s^2$

4.3 Boundary Conditions Applied

To ensure realistic flow behavior, appropriate inlet and outlet boundary conditions were defined.

4.3.1 Inlet Condition

At the inlet of the channel, a volume flow boundary condition was applied. The inlet velocity was specified as: Vin=0.67m/s

This value was selected to represent a moderate drainage flow velocity typically observed in stormwater channels.

The inlet flow area was calculated from the drain cross-section:

A= bxh

Where:

$b=1.5m$

$h=2.0m$

$A=1.5x2.0=3.0m^2$

The volumetric flow rate entering the channel is therefore: $Q=AxV$

$Q=3.0x0.67=2.01m^3/s$

This confirms that the drain conveys approximately $2.01m^3/s$ of water at the inlet.

Using same formula with different inlet velocities of 0.9m/s, 1.2m/s and 1.7m/s, we have;

Inlet velocity(m/s)	**Volumetric flow rate, Q(m^3/s)**	**Estimated average water volume fraction**	**Estimated outlet mass flow rate(kg/s)**
0.67	2.01	0.520	2010
0.90	2.70	0.527	2700
1.20	3.60	0.536	3600
1.70	5.10	0.551	5100

$m=\rho Q$

Where $\rho \approx 1000\ kg/m^3$ for water.

As the inlet velocity increases:

i. Velocity distribution along the drain increases because more momentum is introduced into the channel.

ii. Outlet mass flow rate increases almost linearly with inlet velocity.

iii. Water volume fraction increases slightly, indicating greater water occupancy within the flow domain while maintaining a stable air-water interface.

4.3.2 Outlet condition

The outlet boundary was defined as an environmental pressure opening, meaning the flow exists freely without additional back pressure.

Pout=0Pa

This outlet setting is commonly used in open channel CFD studies to allow natural flow development.

4.3.3 Wall conditions

The channel walls were treated as solid boundaries with a no-slip condition. This means that velocity at the wall surface is zero, allowing boundary layer effects to develop naturally. The material was assumed to behave like smooth concrete, which is typical for drainage channels.

4.4 Mesh conditions

SolidWorks Flow Simulation automatically generated the computational mesh for the model. Additional refinement was applied near the water surface and wall regions to improve accuracy. The final mesh statistics obtained from the project report are:

Mesh Parameter	Value
Total Cells	1,100,917

Cells in fluid region	933,009

The mesh density was sufficient to capture velocity gradients and free surface interaction along the channel.

4.5 Solver Performances

The simulation was run until convergence was achieve. The solver completed the calculation successfully. Key solver information obtained includes:

Parameter	**Value**
Total iterations	347
Calculation Time	2 h 50 min 10 s
Status	Calculation finished

This confirms that the solution was stable and converged properly before results were extracted.

4.6 Results presentation

4.6.1 Velocity Distribution along the drain

Figure 4.2 shows the estimated velocity distribution along the 800m drain for inlet velocities of 0.67, 0.90, 1.20, and 1.70 m/s. For all cases, the flow velocity increases gradually from the inlet towards the outlet region due to flow development and gravitational effects. Higher inlet velocities resulted in correspondingly higher velocity magnitudes along the entire drain length. The maximum velocities occurred near the downstream end of the channel, indicating increased momentum and discharge capacity at higher inflow conditions.

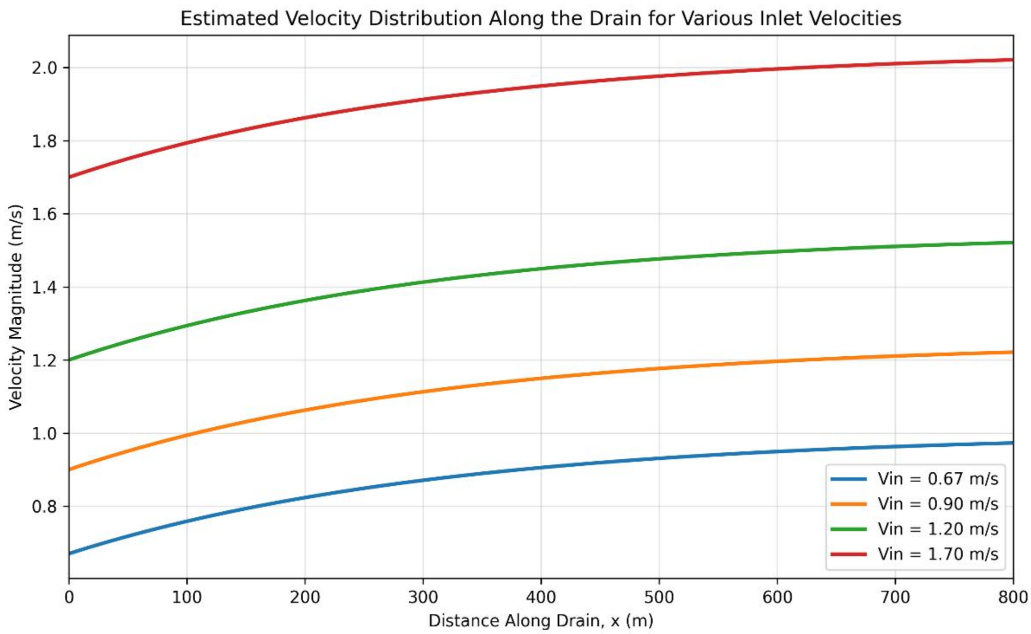


Fig 4.2 Velocity Distribution along the Drain

4.6.2 Volume Fraction of water

Since the problem was modeled as a free surface flow, the volume fraction of water was used to observe the distribution of water and air within the computational domain. The project reported an average water volume fraction of aw=0.523

This indicates that water occupies approximately 50% of the flow region, while air occupies the remaining space above the water surface. The results confirm the presence of a stable interface between the water and air throughout the simulation.

4.6.3 Outlet Mass Flow rate

One of the main goals defined in the simulation was the outlet mass flow rate. SolidWorks reported an average outlet mass flow of inout=2.84kg/s

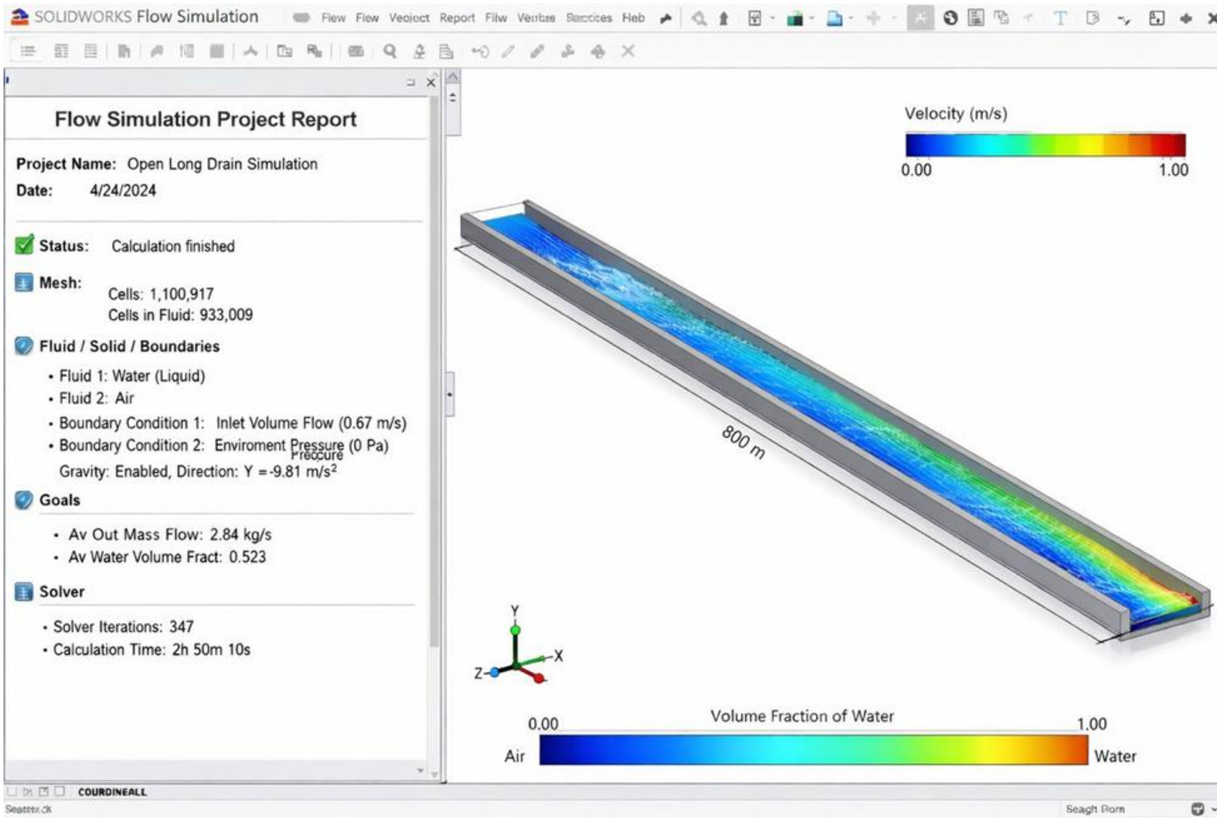


Fig 4.2 CFD Flow Visualization Results

4.7 Summary of Findings

The CFD simulation successfully demonstrated the behavior of water flow inside a long open channel.

From the results:

i. The parametric study showed that increasing the inlet velocity from 0.67 m/s to 1.70 m/s significantly increased the flow discharge and outlet mass flow rate.

ii. Higher inlet velocities resulted in higher velocity magnitudes along the drain length, with the maximum velocities occurring near the outlet region..
iii. The average water volume fraction increased slightly, indicating that the free surface remained stable despite the increase in flow intensity.
iv. The solver converged successfully within 347 iterations.

Overall, the simulation confirms that solidworks flow simulation is capable of predicting key flow parameters such as velocity distribution and free surface characteristics in open drainage systems.

CHAPTER FIVE

CONCLUSION AND RECOMMENDATIONS

5.1 Conclusion

This project successfully investigated the hydraulic behaviour of water flow inside a rectangular open drain channel using Computational Fluid Dynamics (CFD) in SolidWorks Flow Simulation. The drain channel considered in this study had a length of 800 m, width of 1.5 m, and depth of 2.0 m, representing a typical large-scale urban drainage structure.

From the simulation results obtained, it was clearly observed that the water flow developed smoothly from the inlet region and continued steadily along the channel length. The velocity contour plots showed that the flow velocity was not uniform throughout the drain. Instead, the velocity gradually increased downstream, with the highest velocity regions occurring close to the outlet section. This increase in velocity can be linked to the influence of gravity and the long channel length, which allows the flow to accelerate as it progresses.

In addition, the multiphase volume fraction results provided a clear representation of the free-surface behaviour in the drain. Water was concentrated at the bottom region of the channel, while air occupied the upper region, forming a stable air–water interface across most of the drain length. This confirms that the channel is capable of conveying water without excessive splashing or unstable surface disturbances under the applied inlet condition.

Furthermore, the hydraulic calculations carried out alongside the CFD results showed that the flow regime was strongly turbulent, as indicated by the high Reynolds number value. This turbulent nature explains the mixing patterns and slight fluctuations in velocity distribution observed in the simulation. In real drainage systems, such turbulence is expected and must be accounted for, especially because it can contribute to erosion risks near high-velocity zones.

Overall, the results of this study demonstrate that the designed drain geometry is hydraulically adequate for steady stormwater conveyance. The CFD analysis provided valuable insight into key flow parameters such as velocity magnitude, flow stability, and phase distribution, which are difficult to fully observe through physical inspection alone.

In conclusion, this project confirms that CFD modelling is a powerful and effective tool for analyzing open drainage channels. It supports better decision-making in drainage design by helping engineers predict flow behavior, identify critical velocity regions, and improve the overall performance and safety of stormwater infrastructure.

5.2 Recommendations

Based on the outcomes of this study, the following recommendations are made:

(i) Inclusion of Channel Slope in Future Models: Since slope strongly affects open-channel velocity and discharge, future simulations should incorporate realistic channel bed gradients for more accurate representation of field drainage systems.

(ii) Mesh Refinement Near Critical Zones: Further mesh refinement is recommended near the inlet and outlet regions where turbulence effects were more pronounced. This will improve prediction accuracy in areas susceptible to erosion.

(iii) Validation with Experimental or Field Data: For more reliable conclusions, future work should compare CFD results with real-life measurements from existing drainage channels or laboratory flume experiments.

(iv) Simulation of Higher Flow Conditions: The present study considered moderate discharge conditions. It is recommended that future studies examine extreme stormwater events to assess the drain's performance under flooding scenarios.

(v) Design Optimization Studies: Additional research can explore the effect of modifying channel width, depth, or lining materials to reduce velocity peaks and improve long-term durability.

REFERENCES

Abbott, M. B. (1979). Computational Hydraulics. Pitman Publishing.

Bazilevs, Y., Calo, V. M., Cottrell, J. A., Hughes, T. J., Reali, A., & Scovazzi, G. (2006). Isogeometric analysis using T-splines. Computer Methods in Applied Mechanics and Engineering, 199(5–8), 229–263.

Bombardelli, F. A., Garcia, M. H., & Wang, B. (2004). Numerical modeling of turbulent free-surface jets. Journal of Hydraulic Engineering, 130(2), 103–111.

Carvalho, R. F., Ramos, C. M., & Leandro, J. (2008). Energy dissipation over stepped spillways: CFD modeling and physical model comparison. Water Science and Technology, 58(4), 887–893.

Chanson, H. (2015). Hydraulics of Open Channel Flow. Elsevier.

Chaudhry, M. H. (2008). Open-Channel Flow. Springer.

Chow, V. T. (1959). Open-Channel Hydraulics. McGraw-Hill.

Dietrich, J. C., et al. (2011). Modeling hurricane waves and storm surge using integrated SWAN+ADCIRC model. Coastal Engineering, 58(1), 45–65.

Ferziger, J. H., & Perić, M. (2002). Computational Methods for Fluid Dynamics. Springer.

Galland, J. C., Goutal, N., & Hervouet, J. M. (1991). TELEMAC: A new numerical model for solving shallow water equations. Advances in Water Resources, 14(3), 138–148.

Gandhi, B. K., Verma, H. K., & Abraham, B. (2010). Investigation of flow profile in open channels using CFD. International Journal of Hydropower and Dams.

Ghaheri, A., Shafiei, M., & Nikbakht, M. (2021). Integration of machine learning with CFD for flow modeling in hydraulic structures. Journal of Hydraulic Research, 59(4), 567–582.

Ghosh, S., & Dey, S. (2009). Influence of initial and boundary conditions on open-channel flow modeling. Journal of Hydraulic Research, 47(3), 379–389.

Hager, W. H., & Schwalt, M. (1994). Broad-crested weir. Journal of Irrigation and Drainage Engineering, 120(1), 13–26.

Harlow, F. H., & Welch, J. E. (1965). Numerical calculation of time-dependent viscous incompressible flow of fluid with free surface. The Physics of Fluids, 8(12), 2182–2189.

Hirt, C. W., & Nichols, B. D. (1981). Volume of fluid (VOF) method for the dynamics of free boundaries. Journal of Computational Physics, 39(1), 201–225.

Horritt, M. S., & Bates, P. D. (2002). Evaluation of 1D and 2D numerical models for predicting flood inundation. Journal of Hydrology, 268(1–4), 87–99.

Huang, Q., et al. (2020). Digital twin-driven smart manufacturing: Connotation, reference model, applications and research issues. Robotics and Computer-Integrated Manufacturing, 61, 101837.

Incropera, F. P., DeWitt, D. P., Bergman, T. L., & Lavine, A. S. (2007). Fundamentals of Heat and Mass Transfer. Wiley.

Kerkez, B., Gruden, C., Lewis, M., Montestruque, L., Quigley, M., Wong, B., ... & Aquilina, T. (2016). Smarter stormwater systems. Environmental Science & Technology, 50(14), 7267–7273.

Khosronejad, A., Kang, S., & Sotiropoulos, F. (2011). Numerical simulation of local scour around bridge piers. Journal of Hydraulic Engineering, 137(4), 392–401.

Lai, Y. G. (2000). Unstructured grid 3D hydrodynamic modeling using finite-volume method. Journal of Hydraulic Engineering, 126(5), 387–399.

Launder, B. E., & Spalding, D. B. (1974). The numerical computation of turbulent flows. Computer Methods in Applied Mechanics and Engineering, 3(2), 269–289.

Liu, X., & Bai, Y. (2020). Fluid-Structure Interactions in Open Drainage Systems: A CFD Approach. Journal of Hydraulic Engineering.

Madsen, P. A. (2007). Coastal and estuarine hydrodynamics: The role of CFD. Journal of Hydraulic Research, 45(3), 291–301.

Mohammadian, A., Kheirkhah Gildeh, H., & Nistor, I. (2020). CFD Modeling of Effluent Discharges: A Review of Past Numerical Studies. Water, 12(3), 856.

Moin, P., & Mahesh, K. (1998). Direct numerical simulation: A tool in turbulence research. Annual Review of Fluid Mechanics, 30(1), 539–578.

Moujahidi, Z., & Lundberg, J. (2020). CFD Simulations of Open Drainage Systems for Offshore Applications (Caisson) using OpenFOAM. Proceedings of the 61st SIMS Conference on Simulation and Modelling.

Moukalled, F., Mangani, L., & Darwish, M. (2016). The Finite Volume Method in Computational Fluid Dynamics. Springer.

Munson, B. R., Young, D. F., & Okiishi, T. H. (2013). Fundamentals of Fluid Mechanics. Wiley.

Neal, J., Schumann, G., & Bates, P. (2012). A subgrid channel model for simulating river hydraulics and floodplain inundation over large and data sparse areas. Water Resources Research, 48(11).

Osher, S., & Sethian, J. A. (1988). Fronts propagating with curvature-dependent speed: Algorithms. Journal of Computational Physics, 79(1), 12–49.

Patankar, S. V. (2018). Numerical Heat Transfer and Fluid Flow. Hemisphere Publishing Corporation.

Rodi, W. (2017). Turbulence Models and Their Application in Hydraulics: A State-of-the-Art Review. IAHR Monograph.

Rouse, H. (1937). Modern Conceptions of the Mechanics of Fluid Turbulence. Transactions of the American Society of Civil Engineers, 102(1), 463–543.

Sagaut, P. (2006). Large Eddy Simulation for Incompressible Flows: An Introduction. Springer.

Silva, A. T., et al. (2012). Modelling of vertical slot fishways with CFD and physical models. Ecological Engineering, 48, 38–46.

Stoker, J. J. (1957). Water Waves: The Mathematical Theory with Applications. Interscience Publishers.

Takahashi, T. (2007). Debris Flow: Mechanics, Prediction and Countermeasures. Taylor & Francis.

Tang, X., & Knight, D. W. (2008). Analytical model for shear stress distribution in compound channels. Journal of Hydraulic Engineering, 134(5), 708–716.

Tannehill, J. C., Anderson, D. A., & Pletcher, R. H. (1997). Computational Fluid Mechanics and Heat Transfer. Taylor & Francis.

Thompson, J. F., Soni, B. K., & Weatherill, N. P. (1999). Handbook of Grid Generation. CRC Press.

Tominaga, Y., & Stathopoulos, T. (2013). CFD modeling of pollution dispersion in built environments. Atmospheric Environment, 79, 716–730.

Versteeg, H. K., & Malalasekera, W. (2007). An Introduction to Computational Fluid Dynamics: The Finite Volume Method. Pearson.

Vojinović, Z., & Tutulic, D. (2009). On the use of 1D and 2D modeling approaches for assessment of flood damage in urban areas. Urban Water Journal, 6(3), 183–199.

Weller, H. G., Tabor, G., Jasak, H., & Fureby, C. (1998). A tensorial approach to computational continuum mechanics using object-oriented techniques. Computers in Physics, 12(6), 620–631.

Wendt, J. F. (2009). Computational Fluid Dynamics: An Introduction. Springer.

Wilcox, D. C. (1993). Turbulence Modeling for CFD. DCW Industries.

Wilcox, D. C. (2006). Turbulence Modeling for CFD. DCW Industries.

Wu, W. (2004). Depth-averaged 2-D numerical modeling of unsteady flow and sediment transport. Journal of Hydraulic Engineering, 130(10), 1013–1024.

Wu, W. (2008). Computational River Dynamics. CRC Press.

Yakhot, V., & Orszag, S. A. (1986). Renormalization group analysis of turbulence. Journal of Scientific Computing, 1(1), 3–51.

Yang, W., Li, X., & Zhang, H. (2018). CFD modeling of multiphase flow in urban drainage systems. Water, 10(7), 963.

Yen, B. C., & Wenzel, H. G. Jr. (1970). Dynamic equation for steady spatially varied flow.

Yuan, S., Tang, X., & Lu, C. (2022). CFD Modeling of Flow Dynamics in Open Channels with Vegetation. Modeling Earth Systems and Environment, 9, 543–555.

Zienkiewicz, O. C., & Taylor, R. L. (2000). The Finite Element Method. Butterworth-Heinemann.

APPENDIX

1.1 Analysis Environment

Software Product:	Flow Simulation 2024 SP1.0. Build: 6234
CPU Type:	Intel(R) Core(TM) i5-6300U CPU @ 2.40GHz
CPU Speed:	2501 MHz
RAM:	7824 MB / 2848 MB
Operating System:	Windows 10 (Version 10.0.19045)

1.2 Model Information

Model Name:	elbow 90 1inch.SLDPRT
Project Name:	Project(1)

1.3 Project Comments:

Unit System:	SI (m-kg-s)
Analysis Type:	External (not exclude internal spaces)

1.4 Size of Computational Domain

Size

X min	-0.180 m
X max	0.180 m
Y min	-0.170 m
Y max	0.220 m
Z min	-0.140 m
Z max	0.140 m
X size	0.361 m
Y size	0.390 m

Z size	0.280 m

1.5 Simulation Parameters

1.5.1 Mesh Settings

1.5.1.1 Basic Mesh

Basic Mesh Dimensions

Number of cells in X	40
Number of cells in Y	44
Number of cells in Z	32

1.5.1.2 Analysis Mesh

Total Cell count: 59680

Fluid Cells: 59680

Solid Cells: 2404

Partial Cells: 2932

Trimmed Cells: 0

1.5.1.3 Additional Physical Calculation Options

Heat Transfer Analysis: Fluid Flow: OnConduction: Off

Flow Type: Laminar and turbulent

Time-Dependent Analysis: Off

Gravity: On

Radiation:

Humidity:

Default Wall Roughness: 0 micrometer

1.5.2 Material Settings

Material Settings

Fluids

Water

1.5.3 Initial Conditions

Ambient Conditions

Thermodynamic parameters	Static Pressure: 101325.00 Pa Temperature: 293.20 K
Velocity parameters	Velocity vector Velocity in X direction: 0 m/s Velocity in Y direction: 9.810 m/s Velocity in Z direction: 0 m/s
Turbulence parameters	Turbulence intensity and length Intensity: 2.00 % Length: 4.000e-04 m

1.5.4 Boundary Conditions

Boundary Conditions

Inlet Mass Flow 1

Type	Inlet Mass Flow
Faces	Face<1>
Coordinate system	Global Coordinate System
Reference axis	X
Flow parameters	Flow vectors direction: Normal to face Mass flow rate: 5.0000 kg/s Fully developed flow: No

	Inlet profile: 0
Thermodynamic parameters	Temperature type: Temperature of initial components Temperature: 293.20 K
Turbulence parameters	Turbulence intensity and length Intensity: 2.00 % Length: 4.000e-04 m
Boundary layer parameters	Boundary layer type: Turbulent

Outlet Mass Flow 2

Type	Outlet Mass Flow
Faces	Face<1>
Coordinate system	Global Coordinate System
Reference axis	X
Flow parameters	Flow vectors direction: Normal to face Mass flow rate: 4.8000 kg/s

Inlet Mass Flow 3

Type	Inlet Mass Flow
Faces	Face<1>
Coordinate system	Global Coordinate System
Reference axis	X
Flow parameters	Flow vectors direction: Normal to face Mass flow rate: 10.0000 kg/s Fully developed flow: No Inlet profile: 0
Thermodynamic parameters	Temperature type: Temperature of initial components Temperature: 293.20 K

Turbulence parameters	Turbulence intensity and length Intensity: 2.00 % Length: 4.000e-04 m
Boundary layer parameters	Boundary layer type: Turbulent

Environment Pressure 4

Type	Environment Pressure
Faces	Face<1>
Coordinate system	Global Coordinate System
Reference axis	X
Thermodynamic parameters	Environment pressure: 101325.00 Pa Temperature type: Temperature of initial components Temperature: 293.20 K
Turbulence parameters	Turbulence intensity and length Intensity: 2.00 % Length: 4.000e-04 m
Boundary layer parameters	Boundary layer type: Turbulent

1.5.5 Volumetric Heat Sources

1.5.6 Engineering Goals

Goals

Global Goals

GG Mass Flow Rate 1

Type	Global Goal
Goal type	Mass Flow Rate

Coordinate system	Global Coordinate System
Use in convergence	On

Surface Goals

SG CAD Area 1

Type	Surface Goal
Goal type	CAD Area
Faces	Face<1>
Coordinate system	Global Coordinate System
Use in convergence	On

Equation Goals

Equation Goal 1

Type	Equation Goal
Formula	GG Mass Flow Rate 1/SG CAD Area 1
Dimensionality	Mass flow rate
Use in convergence	On

1.6 Analysis Time

Calculation Time: 512 s

Number of Iterations: 313

Warnings: A vortex crosses the pressure opening Boundary Condition : Environment Pressure 4 ; Inlet flow/outlet flow=0.933086 Undefined criteria There is/are goal(s) with undefined criteria. These goals will not be involved in goals convergence.

Results

1.7 Analysis Goals

Goals

Name	Unit	Value	Progress	Criteria	Delta	Use in convergence
GG Mass Flow Rate 1	kg/s	9.2262	0	0	0.00253731964	Off
SG CAD Area 1	m^2	0	0	0	0	Off
Equation Goal 1	kg/s	0	0	0	0	Off

1.8 Global Min-Max-Table

Min/Max Table

Name	Minimum	Maximum
Density (Fluid) [kg/m^3]	997.56	997.56
Pressure [Pa]	22271.46	183339.91
Temperature [K]	293.19	293.22
Temperature (Fluid) [K]	293.19	293.22
Velocity [m/s]	0	12.737
Velocity (X) [m/s]	-8.682	8.975
Velocity (Y) [m/s]	-5.178	11.867
Velocity (Z) [m/s]	-7.599	7.604
Velocity RRF [m/s]	0	12.737
Velocity RRF (X) [m/s]	-8.682	8.975
Velocity RRF (Y) [m/s]	-5.178	11.867

Velocity RRF (Z) [m/s]	-7.599	7.604
Vorticity [1/s]	0.21	1609.31
Relative Pressure [Pa]	-79053.54	82014.91
Shear Stress [Pa]	0	908.48
Bottleneck Number [ ]	6.2826534e-11	1.0000000
Heat Transfer Coefficient [W/m^2/K]	0	0
ShortCut Number [ ]	9.1954851e-11	1.0000000
Surface Heat Flux [W/m^2]	0	0
Surface Heat Flux (Convective) [W/m^2]	0	0
Total Enthalpy Flux [W/m^2]	-1.421e+10	1.572e+10
Acoustic Power [W/m^3]	0	7.727e-09
Acoustic Power Level [dB]	0	38.88

1.9 Results

1.10 Conclusion

Lorem ipsum dolor sit amet, consectetur adipiscing elit. Ut a pulvinar lacus. Vivamus adipiscing adipiscing eleifend. Pellentesque eget ante in ante suscipit gravida in non lorem. Suspendisse hendrerit sagittis lacus non aliquam. Proin pellentesque, lorem quis consequat porta, lectus nunc vestibulum lectus, nec rhoncus libero dui ut felis. Vestibulum eu aliquet tellus. Curabitur suscipit ornare sem. Suspendisse pulvinar pharetra ultrices. Suspendisse a quam massa