CFD ANALYSIS OF OGEE SPILLWAY

A DISSERTATION

SUBMITTED IN FULFILLMENT OF THE REQUIREMENTS

FOR THE AWARD OF THE DEGREE

OF

MASTER OF TECHNOLOGY

IN

HYDRAULICS AND WATER RESOURCES ENGINEERING

Submitted by

Manish Kataria

(2K20/HFE/05)

Under the supervision of

Dr. BHARAT JHAMNANI

DEPARTMENT OF CIVIL ENGINEERING, DELHI TECHNOLOGICAL UNIVERSITY

(Formerly Delhi College of Engineering)
Bawana Road, Delhi-110042

CANDIDATE'S DECLARATION

I, **MANISH KATARIA**, Roll No. **2K20/HFE/05** of **M. Tech (HRE),** hereby declare that the project Dissertation titled "**CFD Analysis of Ogee Spillway**" which is submittedby me to the department Hydraulics and Water Resource Engineering, Delhi Technological University, Delhi in partial fulfilment of the requirement for the award of the degree of Master of Technology, is original and not copied from any source without proper citation. This workhas not previously formed the basis for any Degree, Diploma Associateship, Fellowship or other similar title or recognition.

Place: Delhi

Date: **(Manish Kataria)**

<u>CERTIFICATE</u>

I hereby certify that the project Dissertation titled "**CFD Analysis of Ogee Spillway**" which is submitted by, roll number **2K20/HFE/05** of M. Tech **(HRE),** Delhi Technological University, Delhi in partial fulfilment of the requirement for the award of the degree of Master of Technology, is record of the project carried out by the students under my supervision. To the best of my knowledge this work has not been submitted in part or fully forany Degree or Diploma To this University or elsewhere.

Place: Delhi **(Dr. BHARAT JHAMNANI)**

Date: **(SUPERVISOR)**

ACKNOWLEDGEMENT

I take this opportunity to express my sincere regards and deep gratitude to **Dr. Bharat Jhamnani** (Professor, Civil engineering department, DTU) for his consistent guidance, monitoring and constant encouragement throughout the course of this project work. Also, I express my gratitude to **Department of Civil Engineering, DTU** for giving me such an opportunity to commence this project in the first instance.

Professors and faculties of the Civil engineering, DTU have been very supportive and cooperative. They have been always present for their kind opinions and suggestions regarding this project work and therefore I am deeply obliged to them.

Last but not the least, I would like to thank my family and my colleagues from the department who encouraged me to bring work to a successful close.

Place: Delhi

Date: (MANISH KATARIA)

TABLE OF CONTENTS

ABSTRACT

Computational Fluid Dynamics (CFD) displays are already becoming standard planning and inquiry tools in the design profession. Nowadays, project designs involve the use of CFD methodologies in conjunction with real size demonstrating to break down complicated, rapidly varied, and violent streams that would be difficult to analyse using actual demonstrating alone. In particular, the thinking and use of CFD demonstrating in the subject of hydraulic engineering is on the rise. Apart from being used to test other design approaches, CFD has the potential to become an independent displaying strategy in the design of water-driven structures.

The inquiry begins with details on the real-world design methods to a typical ogee dam spillway. As a result, the calculations and measurements of the real models are current, the challenging methodology and trial outcomes achieved by this showcasing train Ansys-Fluent, a commercially available Computational Fluid Dynamics (CFD) package, was used for CFD visualization. To show the real replica, Reynolds-discovered the middle value of Navier Stokes conditions were combined with the possible k-vortex thickness conclusion model. This theory investigates the connection between CFD model turn of events and its core assumptions. In this reenactment, several test scenarios were assessed, including constant and entirely hydrodynamic situation reproduction for 2d computations to get mostly precise conclusions. The lattice affectability experiment was conducted on the 1d and 3d models to determine the required cross section size. Finally, by contrasting the CFD replica results with the real replica outcomes, the urgent factor interpretation and irrigation levels established by mathematical replica are discussed during an approval conversation. The results showed that the CFD replica can plan the flow of both stream stages since they were similar to those achieved in the original models. Despite the fact that there were a few minor differences in quality, the pictorial pattern remained reasonably consistent throughout all study conclusions.

The Flow Development is seen by the means of Density contours, Pathlines, Velocity vectors.

Pressure Results are seen for different discharges and on different sensors located on the Spillway.

CHAPTER ONE

INTRODUCTION

Dam building has been increasingly important as the world's water supply, flood control, transportation, hydroelectric power generation, fishery, and diversion needs have grown. Because water is so important to ensure, new improvements in dam design and inspection are essential for better irrigation asset management.

As a safety measure against overtopping, all dams are equipped with spillways. When water levels surpass the filled stockpile stage (FSL), they are provided to safely divert irrigation from the repository. In recent years, the frequency of massive floods has increased, resulting in increased inflows into supplies. The basic cause of dam disappointment in South Africa is plainly missing spillway layout, which accounts for 58% of all dam failures. In this method, careful spillway planning is essential to avoid overtopping of the non-excess wall division.

Ogee spillways, which feature a summarised organise block with a S form, have proven to be quite useful. Because of its legal capacity, ability to handle floodwater, and increased security concern, they are regarded the most commonly used spillways. Although the broad ogee form and its stream characteristics are well-known, a little bend in the usual plan naturally affects its stream properties. In any event, these changes to the typical shape require designers to evaluate peak performance.

For more than a century, real exhibition was the only examination tool available, and it was used as the pattern to approve various tactics. Until now, new plan tools have been evolving to analyse rapidly fluctuating stream conditions, thanks to continued developments in computational and mathematical techniques. This advancement has compelled the unavoidable use of mathematical displaying as a common plan instrument in various design controls. This investigation looks into the capability and application of commercial Computational Fluid Dynamics (CFD) programming, notably "ANSYS-FLUENT," to show ogee spillways.

Computational Fluid Dynamics (CFD) is a branch of mathematics that was developed to address problems with liquid streams. Various CFD packages are now available that may be used to show a variety of elastic and non-elastic, laminar and violent liquid flow difficulties.

This technique has been increasingly recognised by the hydraulic barrier designing community as a useful planning tool as well as an informative tool in the exploratory foundations. With the reciprocal use of CFD procedures and actual scale exhibiting in Hydraulic Engineering, it has become increasingly vital to validate CFD displaying using genuine models. Using CFD showing to detect the early problematic stream highlights for model scenarios has also shown to be beneficial.

CFD demonstrating is commonly employed in the spillway stream meadow to break down the diverse pressure driven circumstances, such as atmosphere entrainment, stream division, disturbance, and stun effect. To think about the stream beyond an ogee spillway. Their predictions for the pressing factor commencing outside the spillway and stream tariff were in reasonable agreement with the test results.

The major types used to regulate rising water are ogee spillways, which are erected at the same time as concrete or workmanship dams. They will be considered because of their importance. With the advancement of software engineering and other types of computational liquid elements (CFD) programming, the behaviour of ogee spillways may be addressed in a short amount of time and at a low cost. In this study, FLUENT programming was used to replicate a stream over an ogee spillway. Because the stream over ogee spillway is powerful and has a free surface, its characteristics are often surprising and difficult to predict. This study looks at how different choppiness models predict the water-driven condition of the stream over the ogee spillway. For each circumstance, the volume of liquid (VOF) approach is used to determine the free surface. The results show that when the RNG k-ε model is used, the accuracy of the results obtained from the stream over the ogee spillway is improved.

Spillways are pressure-driven structures that are commonly used in containment and capacity dams to transfer more water and flood during emergency situations. Numerous dams have been destroyed as a result of inadequate and ineffective spillway design. In this approach, studying the flow of water over a spillway is an important design consideration. Stream processing across an ogee peak has recently been done using computational liquid elements (CFD).

The results of the mathematical model that remembered pressure for the spillway peak, water surface profile, and spillway release coefficient were extremely close to the experimental values.

The water-powered design of a spillway and a stilling basin has been one of the most debated topics in water-powered engineering. Stream conditions, spillways, and stilling basin that are properly planned will actually desire to pass flood waters efficiently and safely downstream of dams. For more than a century, actual scale demonstrating has been used in the design and inspection of water-driven structures. A water-driven model is still an accurate device for the test inspection of stream through a spillway structure, but it can only provide reliable data if it

is planned correctly.
With the increase in PC processing power, mathematical simulations of hydrodynamic cycles, such as stream through spillways, are becoming more enticing. For alignment and approval, a comparison of these mathematical outputs with trial or model data is still required. Computational Fluid Dynamics (CFD) is a type of mathematical visualisation created to solve problems with liquid streams. This includes applications that need a strong liquid connection, such as the flow of water in a stream or over and around hydrodynamic structures. As a result, there is a lot of interest among pressure-driven professionals in the usefulness of CFD to display liquid streams.
Spillways and other flood exits are designed to safely transfer floodwaters to the stream downstream of the dam and prevent the dam from overflowing. The water-driven behaviour of spillway constructions is extremely difficult to inspect and decipher. The major types used to regulate rising water are ogee spillways, which are erected at the same time as concrete or workmanship dams. [US Design Standards, 2010] Spillways are powerfully measured to safely pass floods comparable to or not exactly the present basic Probable Maximum Flood (PMF). The conduct of ogee spillways may be concentrated in a short time frame and without paying significant expenses thanks to advances in software engineering and various types of computational liquid elements (CFD) programming. FLUENT programming was used to reconstruct a stream through an ogee spillway to consider pressure variations, and the results were compared to trial results.
In this topic, several experiments based on CFD showing are ongoing. At the guide mass, CFD demonstrating is used to simulate Spillway's methodological channel.
The arithmetic of the guide divider (left) generated the unsteadiness in the stream example and optional vortex stream approaching the start of the approach channel, according to the Kamal Saleh Dam model by flow3d. They observed that the form of the weir reduced its exhibition to remove the peak flood release, and that the flow of water in the spillway is strongly impacted by stream design development.
Fluent requires the space where the liquid stream occurs to be discretized into smaller control volumes before solving the liquid stream concerns. For calculating the liquid stream area, the product uses a plan specific meeting. This area math should then be used to create a framework of triangles or quadrilaterals, which might be structured or unstructured. After completing the pre-planning stage, the familiar layout and solver stage is required before recreation. In the arrangement stage, material properties, limit conditions, and other details are specified, and in the solver part arrangement technique, computing action and stream time may be specified.

The outcomes of the familiar reenactment can be verified in the post-handling phase.

1.1 CFD Modelling

Computational liquid elements demonstrate liquid mechanics standards by employing mathematical methodologies and computations to address challenges with liquid streams. To get a three-dimensional understanding of kettle execution, models can combine material responses—burning cycles—with liquid streams. Where the facade is distinguished by bound circumstances, CFD models attempt to reproduce the assistance of solution and gases. They also keep track of how solids move through a framework. The Navier-Stokes conditions are applied in these models. The recreations are then directed by iteratively addressing the conditions as a consistent state or a transient condition. Because cold stream and CFD demonstrating have now dominated the area, and because they have extensive explicit applications in fuel mixing and the conduct of fuel mixtures, they are discussed in greater depth in the following sections. A great number of the mix options are based on actual exhibiting and CFD demonstrating.

CFD demonstrating provides important information about health in the dispersion of hazardous gas under complicated settings, whether in open air, such as in urban regions with various obstructions and building structures, or indoors, with complex stream heading. There are a number of challenges associated with this type of presenting, including the need for precise display of complex computations, the type of dangerous compounds, supply sources, and natural disaster elements, such as the relationship between hazardous materials flow and blockages, possible interactions with barometrical gases or moisture, the risk of accumulation, and dispersing under extreme temperature conditions or under the effects of various occurrences.

1.2 Analysis of CFD Modeling

CFD stands for 'computational liquid elements,' and it is a branch of liquid mechanics that uses computers to break down the behaviour of liquids and physical structures. As the difficulty of applying material science laws directly to real-world conditions to produce scientific expectancies grew, CFD displaying and examination became a well-known online reproduction arrangement. This was especially true when it came to designing challenges involving liquid flow and heat transfer.

This is where mathematical research and personal computers intersect; online recreation. CFD converts entire differential circumstances into frameworks of straight conditions using mathematical approximations, which are subsequently addressed to provide field esteems such as speeds, pressing factors, and temperatures on a small (but frequently large) number of focuses in the problem space.

Despite the fact that mathematical techniques for obtaining estimated answers for differential conditions have existed for a long time, it is the ability of computers to store large amounts of mathematical data and perform quick operations on it that has transformed innovation into the most useful tool for physicists and designers. Simultaneously, this indicates that the use of CFD to acceptable problems is frequently limited by the processing power available. Due to its flexibility in mathematically addressing conditions of state and real conduct, transmitted in differential or express structure, CFD evaluation addresses the showing of liquid. CFD showing investigations as heat transfer concerns are also of critical reasonable value; each competent reproduction instrument includes modules to determine temperature dispersions near by pressing factor and speeds. A few applications can also include solids testing for flexible disfigurement or compound reactions, as well as other non-liquid applications.

• Travel via channeling and embellishments such as valves, tees, and reductions to anticipate pressure declines, speeds, and vortex formation.

• Vehicles with streamlined characteristics, such as automobiles and aeroplanes, to anticipate drag, lift, and downforce.

• Wind engineering for structures and wind testing to predict wind speeds, vortex formation, and pedestrian safety.

• Central air frameworks, for monitoring channel exposition or advancing warm solace for fake or normal ventilation and energy utilisation.

• Warmth exchangers, to predict heat movement and pressing factor reductions.

• Gadget cooling, to anticipate the use of regular and limited cooling solutions.

• Windmills, in order to predict cutting edge lift, speed, and force age at various breeze speeds.

• Cleanroom layout with contamination dispersal and airborne decontamination control.

• Hydrodynamic execution of the boat and seaward structures.

1.3 Problems of CFD Modeling

The problem is defined by a closed computation, referred to as the 'space' contained by its 'boundary.'

The miracles to be imitated are obvious, such as the existence of warmth, a violent stream, complex responses, distinct phases, varied bodies, and so on, all of which have known material properties and coefficients for state circumstances.

There are recognized introductory characteristics, as well as traits on the boundaries of cognition concerning disciplines. This might include things like stream rates, dividers, temperatures, and heat sources, among other things.

The area's math is divided into 'cells,' which are little fundamental forms. The 'network' refers to the arrangement of all cells. The size of the cells will determine the precision of the arrangement (the smaller the better), but the number used will determine the need for PC memory (the more modest cells, the higher tally, the more memory will be devoured, the more drawn out time the arrangement interaction will take).

1.4 Ansys Fluent

Ansys, Inc. is a Canonsburg, Pennsylvania-based American corporation. It creates and distributes multi physics planning entertainment programming for item setup, testing, and movement, and sells its products and services to consumers all around the world. John Swanson founded Ansys in 1970. Swanson offered financial backers his advantage in the association in 1993. In 1996, Ansys made its debut on NASDAQ. Ansys purchased various other planning plan associations in the 2000s, gaining additional development for fluid components, contraptions planning, and distinct actual scientific examination. On December 23, 2019, Ansys became a segment of the NASDAQ-100 record. John Swanson came up with the idea for Ansys while working at the Westinghouse Astro nuclear Laboratory in the 1960s. Engineers were still doing restricted part inspection (FEA) by hand at the time. Swanson's goal to automate FEA was thwarted by Westinghouse's decision to make all-around useful planning programming, therefore he quit the company in 1969 to pursue the item alone. The next year, he founded Ansys under the name Swanson Analysis Systems Inc. (SASI), operating out of his farm in Pittsburgh.

Swanson employed an included worker PC that was rented on a regular basis to nurture the secret Ansys programming on punch cards. Swanson was hired by Westinghouse as a consultant on the proviso that any code he wrote for Westinghouse might also be used for the Ansys product offering. Westinghouse became an important Ansys client in the same way.

1.5 Features of Ansys Fluent

a) Meshes, Numerics and Parallel Processing

Formless lattice innovation is used in Ansys Fluent programming. For 2-D reenactments, the cross section can be made out of quadrilaterals and triangles, and for 3-D reenactments, hexahedra, tetrahedra, polyhedra, crystals, and pyramids. Complex numerics ensure precise results on a variety of cross section types, including networks with execution hubs and non-coordinating networks with network boundaries. The solvers in Ansys Fluent sprint aggressively and effectively for every actual model and stream type, whether it's a constant situation or a transitory one and incompressible at hypersonic speeds. On Windows, Linux, and UNIX stages, advanced equal preparation capacities may be used to perform reproductions on multi-center processors with distinct processors on a single mechanism and various equipment on an organisation. Ansys Fluent can conduct equivalent estimations on networks with a billion cubicles or more, thanks to 64-cycle innovation. Progressed active burden adjustment organically rearranges processor estimates to maximise efficiency. Ansys Fluent ensures that CFD estimations benefit from the increased processing capacity, regardless of the number of processors used for equivalent computation, ranging from 2 to at least 1024.

b) Dynamic & Moving Mesh

Ansys Fluent's unique lattice capability handles claim testing, remembering for chamber streams, valves, and storage partition. Depending on the scenario, a variety of lattice altering plans, such as layering, even, and remeshing, can be used for various touching fractions within a single recreation. All that is necessary is the underlying cross section and a representation of the limit development. For use with unrestricted movement, amass division, transport hydrodynamics, rocket dispatch, and boiler sloshing, a built-in six-levels-of-opportunity solution is also available. Ansys Fluent's setup of splash separation and burning replica, as well as multiphase replica counting those with the anticipation of complementary surface forecast and compressible stream, make dynamic cross section possible. Ansys Fluent also includes sliding lattice and a number of reference outline models for blending tanks, siphons, and turbomachinery with a proven track record.

c) Turbulence & Acoustics

Ansys Fluent has an unequaled selection of choppiness models, including several variations of the well-known k-epsilon replica, k-omega replica, and Reynolds pressure replica (RSM). Ongoing advancements in disturbance displaying have aided in the execution of extra replicas such as violent change replicas, which are useful for demonstrating the transition from laminar to fierce stream that occurs near limits, and a Scale Adaptive Simulation (SAS) choppiness model (beta usefulness), which gives a consistent arrangement in stable stream locales while settling choppiness in transient hazards such as huge partition zones, without an express framework. The enormous whirlpool reenactment (LES) replica and the more efficient confined swirl recreation (DES) model have become enticing choices for mechanical replicas as PC power has increased and processing costs have decreased. For acoustics, ANSYS FLUENT may multiple times detect the commotion caused by risky pressuring factor variations. Using the built-in Fast Fourier Transform (FFT) equipment, fleeting LES predictions for external pressing factor may be converted to a recurrence range. The Fflowcs-Williams and Hawkings acoustics relationship may be used to show the formation of acoustics hotspots for a variety of objects, ranging from presenting solid bodies to rotating fan sharp edges. Broadband commotion source models allow for the evaluation of acoustic sources based on the outcomes of consistent state reenactments.

d) Heat Transfer, Phase Change & Radiation

Warmth transfer is possible with a variety of liquid stream processes, and Ansys Fluent offers a comprehensive range of options for convection, transmission, and emission. The P1 and Rosseland replicas are available for optically thick, taking an interest medium, and the examination reason support aircraft to plane replica for non-participating medium. Separate ordinates (DO) replica is also available and suitable for any intermediate, counting beaker. Additionally, for environment manage reproductions, a sunlight-based load model and a double warmth exchanger replica are available.

Other capabilities closely related to heat transfer include cavitation replica, compressible fluids, covering transmission, real talk, and soaking vapour.

e) Reacting Flow

Since its inception, substance response demonstrating has been a hallmark of Ansys Fluent programming, particularly under tumultuous situations. Ansys Fluent makes use of more recent models such as swirl scattering, PDF transport, and solid restricted rate scientific models, as well as mature models such as whirlpool dissemination, harmony blend section, flamelet, and premixed burning replica. In-situ versatile organisation (ISAT) may be used in conjunction with the EDC or PDF bring replica and increases the speed of tempestuous restricted rate science by a large amount or more. Ansys Fluent's standard reacting stream replica may be used to manage a wide range of vaporous, coal, and fluid fuel burning replicas. There are also excellent replicas for forecasting SOx growth and NOx organisation and annihilation. Outside response capability in Ansys Fluent considers reactions among gas and surface species, as well as between different species, with the purpose of fully anticipating affidavit and drawing. The response models in Ansys Fluent may also be used in conjunction with the real gas model, as well as the LES and DES disturbance models.

f) Multiphase

Ansys Fluent is a multiphase displaying innovation pioneer. Its many capabilities enable designers to gain insight into technology that is sometimes difficult to test. Ansys Fluent employs the Eulerian multiphase replica with its many liquid condition configurations for interpenetrating liquids or stages, as well as a more conservative combination model. Both approaches are capable of dealing with granular streams. Ansys Fluent also supports a number of different multiphase models. The distinct stage replica (DPM) may be used for several multiphase requests, such as splash dryers, fluid fuel showers, continual fibre picture, and petroleum heaters. Where the prediction of the boundary is of interest, the degree of liquid replica is accessible with the anticipation of complementary surface streams, such as sea waves. The cavitation replica has shown to be effective in demonstrating hydrofoils, syphons, and fuel injectors. A few population balance replicas are also available for displaying size distributions.

g) Post-Processing and Data Export

The post-handling tools from Ansys Fluent may be used to create important designs, motions, and reports that make it simple to share CFD findings. Surfaces that are hidden and simple, pathlines, vector plots, form plots, custom field variable definition, and scene development are just a few of the post-processing features available. For further examination, arrangement data can be transferred to Ansys CFD-Post, external illustrations bundles, or CAE bundles. Ansys Fluent arrangement information may be intended to Ansys reenactment outside for use as a warm or pressing factor many in the Ansys Workbench climate. Ansys Fluent may also plan main and warm many on the outside and hotness in quantity from Ansys Fluent to outer FEA networks in its standalone form.

h) Customized Tools

Client-defined capabilities are a popular option for clients that need to customize Ansys Fluent. Complete documentation and a variety of educational activities are available, as well as comprehensive specialist assistance. The Ansys global consulting team may provide or assist in the creation of formats for any gear that has been rehashed. Extra modules, such as PEM and strong oxide power devices, and magnet to hydrodynamics, are available for some unusual applications.

1.6 Spillway Classification

Spillways can be arranged based on a variety of factors: Function (Administrative spillway and helper spillways); administrative or manage organization (Gated spillway, ungated spillways, and conduit hole spillway), with the latter being the most appropriate aspect.

Overflow or Ogee Spillways as mentioned in the preceding section, there are several types of spillways now in use; however, only the unconstrained ogee-molded spillways will be investigated in this theory, as the others are outside the scope of this proposal.

The forms, in fact, are based on a basic parabola that is restricted in order to maintain the orientation of the inferior nappe. To aid the page on the substance of the flood and stream up to the cover of a still basin or into the spillway release canal, the shape under the higher bend is postponed extraneously, down the incline.

Streams above the peak should attach to the outline's material to prevent air from entering beneath the item. The stream skims over the crest without stopping for releases with a designed head.

Outside barrier is removed, and the most extreme release efficacy is achieved.

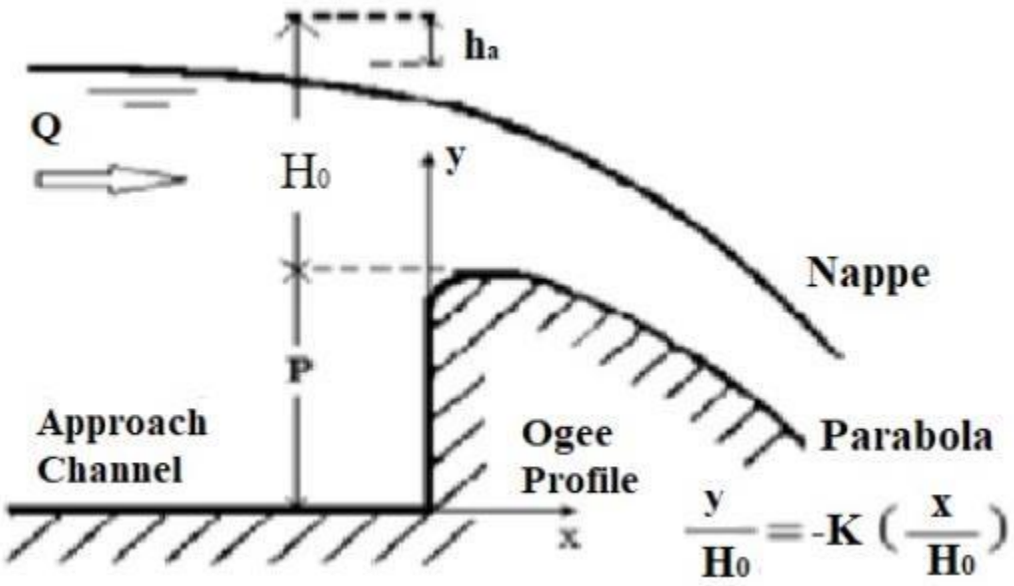


Figure 1: Ogee spillway overview

The Ogee spillway is the most common kind, especially in high-barrier applications. Its ability to bypass streams efficiently and safely when properly planned, along with reasonably good stream prediction abilities, has enabled architects to use it in a variety of situations.

Three distinct plans for spillway management are acknowledged and may be called based on the administrative or manage arrangement: unrestrained gadgets that do not utilise a water powered entrance in their activity, portable peak gadgets, and regulating gadgets.

Uncontrolled peaks are most commonly used on small spillways and weirs, where the arrival of hose is only required if the repository head exceeds the design level. One of the advantages of this design is that it does not require the constant supervision of an administrator, as well as upkeep and repair charges.

When there is an appropriately extended uncontrolled peak or when the spillway peak is located below the repository's standard operating height, versatile peak and directing gadgets are typically used.

1.7 Discharge computations determinations

In light of its form, an ogee spillway is defined by a usually high release coefficient. Regardless, this coefficient isn't constant. The depth of the methodology, the link of the true peak form to the perfect nappe shape, the upstream characteristics, tilt downstream cover boundary, and downstream submergence are all factors that influence it.

For ogee weirs with an upward upstream face, the plan release coefficients. As a result, these coefficients are significant only when the ogee confirms the perfect nappe shape; that is, when $He/Ho = 1$. When the peak forms do not match the precise form, or when a head that is more prominent or modest than the plan head has been completed

The spillway's upstream sloping face causes the coefficient of release to change. The tilted upstream visage roots an increase in the release coefficient for small proportions of the technique profundity to the head on the peak, specifically for statures greater than the plan head.

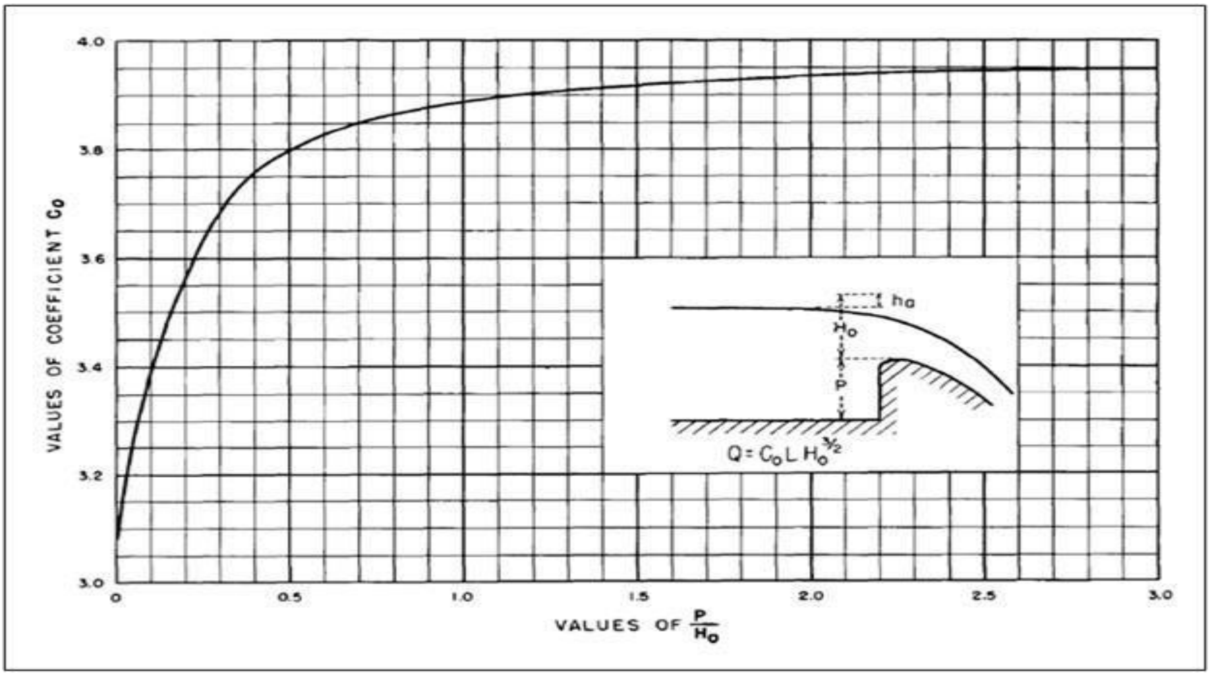


Figure 2: Release coefficient for perpendicular face ogee peak

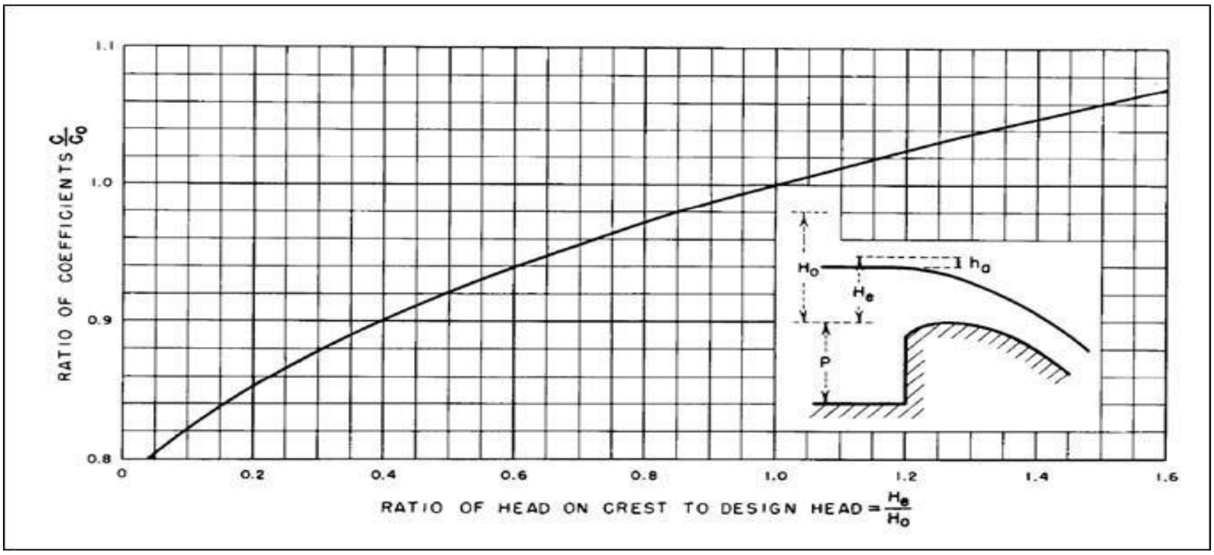


Figure 3: Release coefficients for other than the plan head

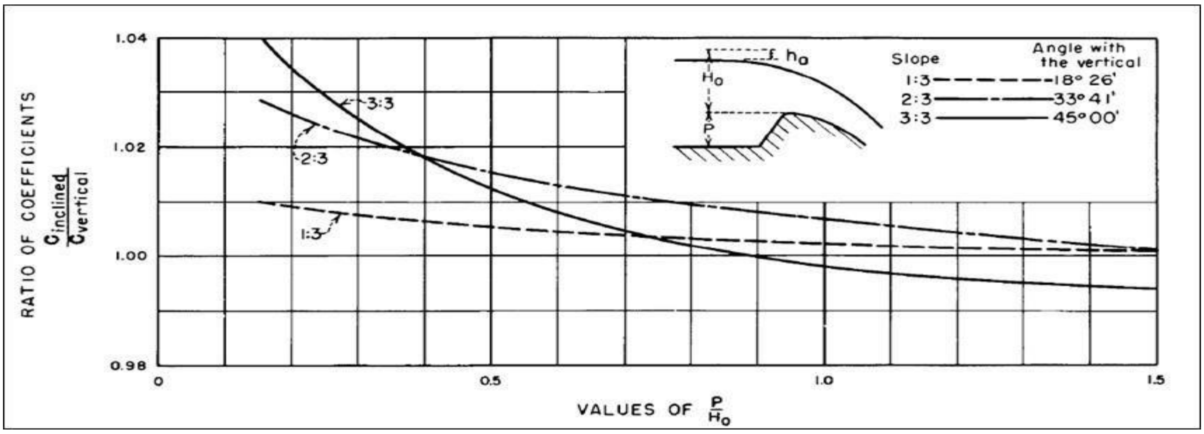


Figure 4: Release coefficients for ogee-shaped summit with slanting upstream features

1.8 Pressure distribution

In terms of operational requirements, underlying strength, and economy, the plan head is often chosen to provide the highest pressure-driven productivity. Ogee peaks operate with the plan release coefficient "Co" to depict air pressures at the plan head "Ho."

The coefficients of release "C" are not exactly the plan coefficients of release "Co," and hopeful pressing factors develop on the peak for heads "He." For heads that are more notable than the plan head, the coefficient of release "C" is more significant than the plan coefficient of release, with unconstructive tension at the top, thereby widening the release limit.Peak negative pressuring factors should not be allowed to become excessively negative, since they might cause cavitation damage, spillway destabilising, and possible design disappointment.

In terms of operational requirements, underlying strength, and economy, the plan head is often chosen to provide the highest pressure-driven productivity. Ogee peaks work with the plan release coefficient "Co" at the plan head "Ho," displaying the climatic pressing variables.

For heads "He," the coefficients of release "C" are not exactly the plan coefficients of release "Co," and hopeful pressuring circumstances develop on the peak. The coefficient of release "C" becomes more prominent than the plan coefficient of release for heads larger than the plan cranium, with unhelpful stress on the peak, hence widening the release limit. Peak negative pressuring factors should not be allowed to become very negative, since they cause cavitation damage, spillway instability, and even the entire structure's failure.

1.9 Governing equations of CFD modeling

Where the challenges are truly stated by a set of halfway differential circumstances, the key standards for all mathematical models remain comparable. Similarly, CFD techniques are governed by a number of factors that must be handled in every manageable quantity.

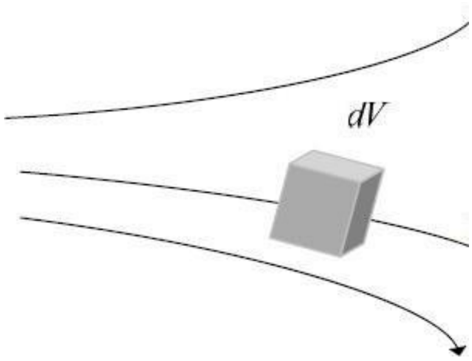
The genuine marvels are addressed by numerical proclamations that are alluded to as administering circumstances of liquid stream and warmth motion, depending on the possessions of liquid stream to imitate. Mass conservation or progression, force and energy conditions, often known as the Navier-Stokes conditions, are included in these criteria. The following diagram depicts mass conservation and force circumstances in light of Ronis' idea.:

a) Mass conservation (continuity) equation

The mass conservation condition, also known as the coherence condition, states that the collection of a closed arrangement of essence will remain constant regardless of the cycles operating inside the framework. Regardless of whether the form or amount of the liquid stream changes as it moves, it has a fixed accumulation.

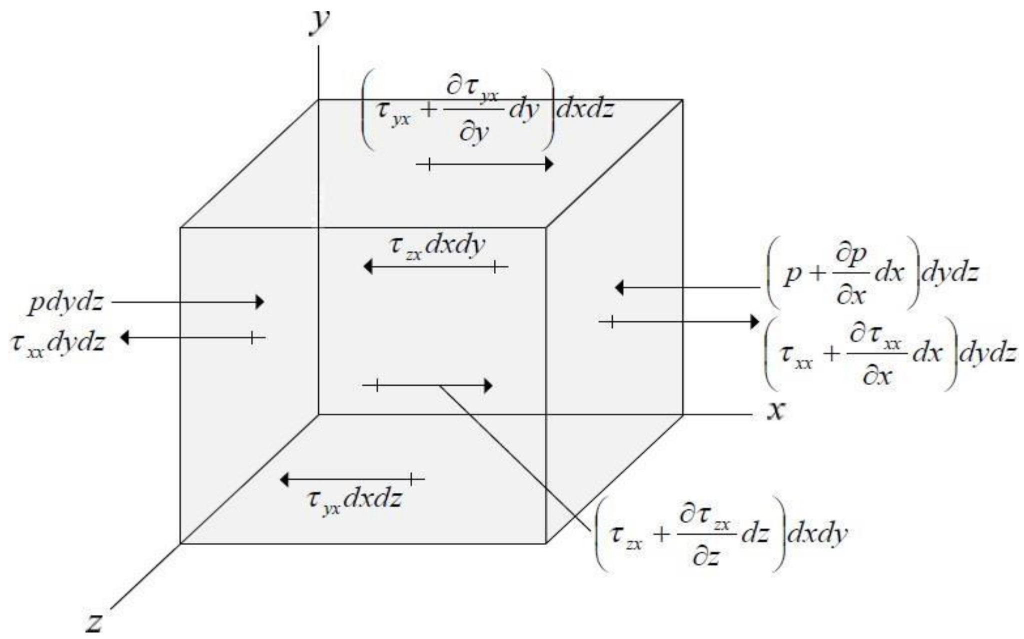
The representation is complete in Equation (2-10) :

$$dm = \rho dV$$



b) Momentum conservation equation

The energy condition is a statement of Newton's Second Law that describes the quantity of powers acting on a molecule of liquid. Model of a flowing liquid component is outlined with more subtleties in Figure underneath.



1.10 Multiphase modeling

Multiphase modeling is a technique for reenacting a stream in the presence of many phases. Each of the three stages (gas, fluid, and strong) is distinguished by a distinct inertial reaction to a cooperation with the stream and the probable field. Multiphase streams are divided into four categories: gas-fluid, gas-strong, fluid-strong, and three-stage streams. The availability of two broad approaches for mathematical estimating, in particular Euler-Euler (degree of Fluid replica or VOF replica) and Euler-Lagrange (separate stage replica), has made such mind-boggling frameworks possible.

Quantity of liquid replica: The stated quantity of liquid replica is designed for at least two immiscible liquids when the location of the liquids' border is important. It's a free outside stream boundary capturing scheme in which the boundary of each liquid is the point of primary attention. The stages are handled with separately in the VOF model, and each stage has its own set of protective requirements.

The fields for speed, pressing factor, and hotness are completed to be something quite comparable due to the volume portion of each stage all through a similar manage amount.

Scattered stage replica: By including the power stability on the molecule, the dispersed stage model is used to depict discrete subdivisions as they navigate through and interact with liquid stream in reality, by following the movement and calculating the paces of advance of preserved possessions.

Contrasting DPM with VOF demonstrating: The DPM showing programme is more harder to use than VOF. VOF is also useful and efficient in terms of computing. The plan of place vectors as the solids respond to shaving pressure are the source of DPM's load. Actual laws do not have place vectors in VOF plans, and speed appears as the major variable, revealing all liquid stream designs crucial. This is due to the way violent liquids exposed to shear pressure continually change form as the pressure is applied. In any event, the relevance and utility of VOF can be achieved at a high computational lattice objective. The two models can't be used with thickness-based solvers, which is a common drawback; only the pressing factor-based solution is allowed.

1.11 Objectives of study

The main goal of this project is to investigate the mathematical recreation of the ogee spillway's hydrodynamics using actual displaying. This is done for a few different versions reenacted with the Ansys-Fluent CFD package and compared to lab investigation data.

The following are the specific objectives of this investigation:

1. Examine the scientific approaches used to plan an ogee spillway based on USBR, USACE, and other procedures.

2. Complete trial testing on two genuine replicas presented in the research facility waterfall and accept the essential boundaries, such as stream overcharge, pressure appropriation on the peak, and negative pressing factor for releases more than the plan head.

3. To do 2d and 3d CFD simulations of a chosen spillway under consistent and totally hydrodynamic conditions using the same spillway math and stream conditions as those used in the research centre.
4. To determine the precision of the CFD results by comparing them to exploratory results.
5. To determine the pros and drawbacks of using CFD demonstration devices to examine a stream flowing over an ogee spillway.

1.12 Research methodology

The investigation begins with a comprehensive writing lesson to provide a state of knowledge about ogee spillway plan and stream features. Different literature, logical diaries, and news were examined in this case. The scientific procedures used to explore stream more than ogee spillway were nitty gritty in the writing study. In addition, the basic hypothesis used in CFD showing was examined. Two genuine replicas were built up in the research centre cascade for the exploratory work. In this way, two steps have been measured in order to obtain the most productive consequences: A time of transition that featured mathematical research and primer tough, which included a rigorous review of each piece of the anticipated ogee plan, including preliminary to reveal duplicate deformities and insufficiencies. The exploratory information experiment used PC showing to conduct a mathematical inquiry and provide nitty gritty facts on the complete stream system. A correlation with the outcomes obtained from actual models validated the computational demonstration findings. The approval interaction is depicted in Figure

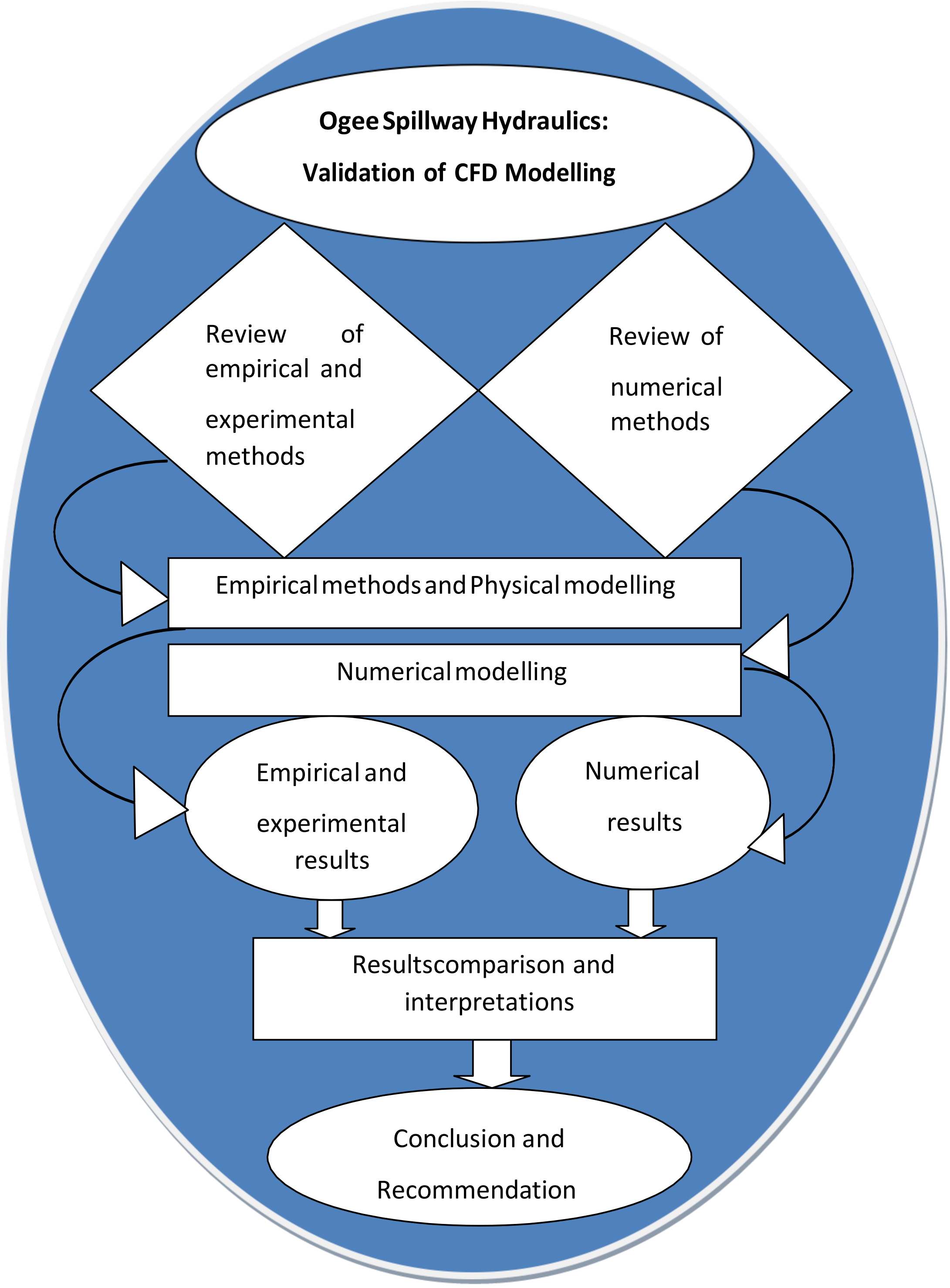


Figure 5: Evaluation of CFD modelling of ogee spillways.

CHAPTER TWO
REVIEW OF LITERATURE

Cassidy (1965) investigated spillways and calculated power scattering using a mathematical approach. Cassidy attempted to determine stress on the peak of an Ogee spillway in two-dimensional space in this study.

Sorensen (1985) considered power dispersal rate In Ogee and ventured spillways. Similarly, power distribution speed in ventured spillways was around 75% faster than in Ogee spillways. Similarly, stilling basin measurements show an eighty-four percent total reduction.

Pegram et al. (1999) commissioned a comprehensive investigation to investigate the influence of mathematical features of steps and size on the kind of stream and power distribution speed, with the goal of achieving equilibrium conditions downstream. When evaluating stream power downstream of the ventured and Ogee spillways, which were dimensionally equivalent, the ventured spillway was shown to have a faster power dissemination speed. Furthermore, the findings show that increasing the inclination of the spillway reduces power distribution.

Tabbara et al. (2005) used ADINA-f programming and a restricted component approach to record the water stream contour across an explored spillway. They also used a standard K-ε replica to determine stream disturbance. The figured irrigation stream profile in relation to all developed ventured spillways was subjectively in knowledge of stream qualities and numerically similar to the intended advantages of free outer stream outline. In addition, related power dispersal was calculated and compared to the test's learned features.

Mansoori and Pedram (2008) considered end ledges ventured spillways and developed a link to determine power scattering speed of these spillways, as discovered. They used a metal skeleton and plexiglass dividers to connect twenty three tiers of plexi goblet (with a width of ten mm, a span of seventeen cm, a breadth of fifty-five cm, and a height of three cm) from the top. They ran their tests with releases of 3.6 litres per second (Nappe stream) and 25 litres per second (Skimming stream), recording the standing pressing factor imposed on the floor and its variations with stream speed and depth. They may establish a link to determine energy dissemination rate in the ledge ventured spillway at Nappe stream system by investigating the collected information and adjusting Rajarantnam and Chamani's condition. Finally, they concluded that the existence of ledge would influence the rate of energy distribution in ventured spillways; however, the impact on the stream is different and is lessened when the discharge is increased. In the Nappe Flow system, energy dispersal rate is influenced by the width, stature, and heavenly messenger of the ledge upstream; in general, increasing the width and tallness of ledges increases power dissemination rate, while decreasing the heavenly messenger of the ledge upstream decreases energy scattering. In this way, vertical ledges outperform bent ledges in terms of power distribution, yet ledges have a little influence on power dispersal in the Skimming stream system due to their misfortune using the FLUENT mathematical model.

Naderi Rad et al. (2009) investigated energy dissipation in several types of ventured spillways, including simple, ledges, and tilted ones. A total of 33 mathematical models were created, including eleven spillway groupings. Three releases and three limit conditions were investigated in these eleven meetings. Five model gathers were measured for each of the fundamental ventured spillways, along with three releases (0.0190, 0.537, and 0.0987 m3/s, spillway plan tallness (hd) of 0.05, 0.1, and 0.15 m, and basic stature (Yc) of 0.0334, 0.0667, and 0.1 m. Three model gathers were considered for each of the ledges and inclined ventured spillways. Three releases (0.0190, 0.0537, and 0.0987 m3/s), spillway plan stature (Hd) of 0.05, 0.1, and 0.15 m, and basic tallness (Yc) of 0.0334, 0.0667, and 0.1 m were explored for each gathering. When the height of the dam remains constant, increasing the number of steps generates an increase in energy dispersion in ventured spillways. When the height of the dam remains constant, decreasing the angle of the ventured spillways produces an increase in energy distribution. When the dam height, number, and amount of steps, as well as the spillway elevation, remain constant, increasing the slant of the steps' floor generates an increase in energy dispersing in the slopped ventured spillways. Expanding release under each of the three circumstances of the ventured spillways causes power dispersal and scattering to decrease, resulting in a free stream.

Mansoori and Soori (2013) investigated energy distribution in the Nappe and Skimming stream systems using FLOW-3D. They dissected newly comprehensive results as well as investigating the identification of stream type in explored spillways. They also looked at a trial link that was established to process energy scattering. The inquiry was predicated on Stephenson's equation. To the point where they compared the general power scattering obtained by FLOW-3D to the value achieved by Stephenson's condition. The mathematical model's and Stephenson's relationship yielded predicted results.

Hamedi et al. (2016) used FLUENT to study four stages (39 to 42) of a 60-venture spillway. The four slant and ledge phases were installed at the same time. Varied ledges of various heights and thicknesses were tested. Invert slants of 7, 10, and 12o were applied in three phases. The energy scattering was measured using a dimensionless barrier m/h. The obtained findings revealed that in a 30 lit/s release (release per unit width q = 0.0225 m2/s), which exhibits the Nappe stream system, the methods with synchronous slant and ledge (where the percentage of m/h is less than 0.7) jointly generate energy dissemination.

Modeling using Reduced Orders (ROM): ROM is an approach for replacing the primary model with a much more straightforward soliciting model that can nonetheless accurately portray essential aspects of an association. The fundamental idea underlying ROM is to identify a decreased cause that has reduced the amount of levels of chance that have strayed from the model's main course of action. Proper Orthogonal Decomposition is the most known method for locating an optimum explanation (POD). A case is an actual model evaluation to determine the most common strategy. We recommend for ROM and POD details. The fluid stream and heat movement were multiplied in and separately. Brenner, T. A., et al. have dealt with multiphase heat transfer concerns employing ROM, where they discovered excellent simultaneousness with the full solicitation m odel, and they have highlighted several practical issues of POD in their study. Lappo, V., Habas hi, W., and Lieu, T., et. al. have used ROM to produce continual enjoyment, with Lieu, T., et. al. demonstrating a whole plane course of action and promising outcomes.

Marker and Cell (MAC) technique: Harlow and Welch introduced the system with a restricted difference staggered structure. Initially, it was intended to settle free surface streams. They employed a marking particle to stamp the fluid-filled cell and used presentation to follow the improvement of the surface. We recommend for subtleties of the primary MAC. In, an improved form of MAC called SMAC was depicted with the assumption of free surface stream generally space. To settle the Navier- Stokes condition, a joint technique employing FEM and MAC was developed. A structure heat move issue had been resolved. McKee, S., et al. performed an incredible composition review for the most part. Furthermore, they have evaluated the latest MAC enhancements.

Smoothed Particle Hydrodynamics (SPH) : SPH strategy is a molecule-based approach developed by Gingold and Monaghan (1977) and Lucy (1977) to address astronomical problems however, it has one of the most well-founded strategies in CFD. Liquids are addressed by a distinct element in the SPH design, and the element's properties are then rounded by bit capabilities over the element within a particular span. This method was used to address a wide range of liquid stream and heat transfer challenges in order to achieve continuous or near-continuous replication with reasonable accuracy. M. B. Liu and G. R. Liu have provided an outstanding overview of recent advancements in SPH technology. SPH has recently proven to be particularly capable of dealing with multiphase and free surface issues. Szewc, K. et al. completed a study on the use of SPH to multiphase streams in.

Complex free surface and multiphase problems were addressed in and, where they show how to use SPH to capture a boundary. Warmth movement concerns were solved with great accuracy. SPH executions were carried out in and to address various CFD difficulties.

Quick Multipole Method (FMM) : L. Greengard and V. Rokhlin established the Quick Multipole Method (FMM) in 1997 as a molecular approach. Cheng, H., et al. proposed a flexible application of FMM to settle Laplace conditions in 3D later in 1999. The multipole development is used by FMM to determine the power between particles. The model is more precise the more terms we have in the development. This method may be used to handle precision and speed. Greengard and Kropinski used FMM to calculate the volume fundamental of incompressible Navier- Stokes conditions and completed execution where is the number of foci in the area discretization. Yokota, R., et al., recently completed a pet scale choppiness near to continuous reproduction using GPU engineering.

Technique for Fundamental Solutions (MFS): Kupradze and Aleksidze proposed MFS in 1964 as a way for dealing with specific elliptic limit esteem issues. The assumed arrangement is given in MFS as a straight mix of primary arrangements. The book covers both the principles and the nuances of this approach. Incompressible Navier-Stokes circumstances and interface concerns were addressed in. Warmth move coefficient was calculated for complicated problems in MFS.

The Limited Pointset Method (FPM) : FPM is a molecular approach for problems involving continuum mechanics, such as liquid streams. It's been tweaked all around to simulate many complex time subordinate streams, moving surface, free surface, and warmth movement concerns. FPM has overcome the re-cross section for time subordinate moving surface streams, which is a major drawback of lattice-based techniques. The Fraunhofer group in Germany has developed a model for reenacting FPM.

Many exciting topics were addressed, such as refuelling the engine vehicle, airbag sending, and so on. Incompressible Navier-Stokes situations and multiphase concerns were also addressed. The use of FPM to heat conduction concerns is discussed.

MPS (Moving Particle Semi-Implicit Method): MPS is a molecular approach developed by Koshizuka and Oka in 1996 for generating incompressible free surface streams. MPS is similar to SPH, however it uses a simpler differential administrator rather than taking angle of part work like SPH. Tokura used LS-DYNA (programming bundle) to compare SPH with MPS and discovered that MPS performs better for certain problems than SPH due to its simplicity. There are various publications available on free surface streams using MPS, many of which are relevant to our benefit.

Quick Fluid Dynamics (FFD) : FFD is an approach for settling Navier-Stokes conditions that falls between between network free and cross section-based techniques. It was invented by Zuo W. and Chen Q. in order to simulate continuous or faster than continuous replay of wind current in buildings. The FFD approach, according to the developers, was informative but less exact than CFD. They used GPU to implement their method, achieving 500-1500 times faster reenactment than CFD on CPU. M. Jin et al. have replicated light-driven streams inside buildings on an extraordinarily large scale. Following that, a few tweaks were made to increase the accuracy of the FFD approach.

PIC (Particle in Cell Method): PIC is a crossover method that uses particles to address the characteristics of liquids. The information is subsequently transferred from particles to a matrix, where the characteristics between particles are inserted. In the mid-1960s, it was a particularly effective tactic in plasma reenactments. In addition, this approach was applied to solve challenging CFD problems such as high-energy atomic crashes in multiphase combination and stun and rarefaction streams. PIC was used to solve a more advanced problem like a fluidized bed.

VIC (Vortex in Cell Method): Vortex method refers to a group of tactics that focus on settling the vorticity rather than the force condition. The problem with the vorticity condition is that it does not directly supply the speed. To obtain the speed field, the Biot-Savart law must be solved, which takes a long time on a computer. Nonetheless, the VIC method can overcome this disadvantage by using a semi-Lagrangian strategy To get the speed field, VIC transfers the vorticity particles to a network and handles the Poisson requirement. Incompressible streams in 3D, disturbance in 2D, and vortex structure in 3D were all seen using VIC.

Cross Section Boltzmann Method (LBM) : LBM is a kind of CFD approach that deals with the Lattice Boltzmann condition rather than the Navier Stokes condition. It depicts a liquid stream using a molecular network technique, in which the particles reside at the nodes of a discrete cross section network. For an introduction to LBM and its use in design. An excellent overview of LBM for single and multiphase streams, as well as the latest LBM advancement. Expansion to a Thermal LBM for multiphase stream and heat transfer. Rosdzimin, A. R. M., et al. addressed the topic of convective heat transfer in, finding advantages of LBM over a conventional Navier Stokes condition in terms of Knudsen number. Low Knudsen number warmth movement difficulties may be reproduced using LBM. LBM was used to solve more sophisticated heat transfer problems, such as the warm conduct of a bead on a strong surface and improved heat transfer for nanofluids. Using LBM, scientists were able to solve complicated multiphase and heat transfer issues recently, as well as complete continuing CFD. For example, Gevelera M. et al. used LBM to replicate a variety of complicated liquid streams and achieved consistent reproductions using multi-center engineering.

G.Satish, K.Ashok Kuma, V.Vara Prasad, and Sk.M.Pasha focused in their study on how the path via sudden and gradual differences in pipe diameter (enlargement and constriction) was mathematically replicated with water by temperamental stream in k-epsilon conspire. During the time spent moving along these lines, the significant perceptions created identified with the pushing factor and speed develops. Unexpected amplification causes stream swirls to be more intense than sudden withdrawal. Furthermore, the misfortunes are concentrated towards the beginning of the line's extension. Vena contracta's are moulded at the point of compression during unexpected constriction, and consistency has no effect on the pressing factor drop after sudden withdrawal.

Wan Kai and Wang Ping's work focus was on Using conventional k-model with FLUENT programming on huge width CFD mathematical reproduction of wind current in a 90° bent cylinder. The whirlpool consistency model, which gets closed RANS conditions to settle the model, has a place with the regular k-model. The encircling air is the used liquid medium, with a thickness of 1.225 kg/m3 and a kinematic consistency of 1.7894e-5 kg/m.s. With a present wind speed of 15 m/s, the elbow moves in a consistent and stable manner. Because the stream rate is so low, it's often mistaken for an incompressible liquid. The control condition administers the Cartesian facilitate framework under adiabatic, consistent, incompressible liquid stream via homogenization of the coherence condition and instantaneous Navier-Stokes conditions.

(US Reclamation Service, 1987) The Ogee-peaked spillway is one of the most significant and consistent pressure-driven structures, and its superior water-driven attributes allow it to release excess water or floods that exceed the capacity volume. In many situations, ogee-peaked spillways are used as water discharge structures.

(Peltier and colleagues, 2018). They are efficient and safe when designed and executed precisely, and they can also accurately assess stream rates. A zero relative pressing factor is often seen along the surface profile when the pressing factor head arrives at a plan head.

(Specialists, 1952) However, an ill-advised building strategy might result in dam failure. Overtopping due to an inadequate spillway plan, garbage obstruction of the spillway, or resolution of the barrier peak account for approximately 34% of all dam failures in the United States of America; hence, these spillways must be meticulously designed to ensure stream quality.

(Savage and Johnson, 2001) Due to its appearance and ability to convey overflow water effectively and safely with sensibly outstanding stream estimation skills, the ogee-peaked spillway is likely the most considered pressure driven design. As a result, engineers employ it in a wide range of situations.

(2015, Kanyabujinja) To prevent air from passing under the water sheet, flow over the Ogee-peaked spillway should stick to the silhouette's material. In terms of head plan, the stream floats more the outer form with low limit shell impedance, resulting in optimal release efficacy.

Peltier and his colleagues (2015) Over an Ogee peaked spillway, there are often few learn on the stream characteristics, especially for heads larger than the plan head. Furthermore, there is insufficient data on the upward speed distribution along the peak profile allowed pressing factor calculations and speed circulations directed in two water powered vehicles. An Ogee spillway replica with several scale components. These were created using head dimensions that are more noticeable than unit.

(Ho et al., 2006; Kim and Park, 2005; Savage and Johnson, 2001; Chanel and Thesis, 2009). In hydrodynamic examinations, such as analyses of stream across Ogee spillways, numerical reenactments are useful. For modification and approval, a connection of these mathematical outputs with exploratory data is still required. Flow-3D, a cost-effective Computational liquid Dynamics programming package, using the restricted volume technique to address challenges such as liquid stream. The computational space is divided into a network of changeable measured hexahedral cells using Cartesian directions. The typical attributes for the stream borders, such as pressing factor and speed, are handled at discrete times for each cell. The majority of research on CFD-based spillway visualisation use Flow 3d, which sets the Reynold's averaged Navier-Stokes (RANS) conditions.

PIV stands for pulse intensity velocity (Adrian, 1991; Sveen and Cowen, 2004). It uses a particle tracer to track liquid movement. The PIV theory is based on a small tracer element estimate. These are insufficient to track the progress of the liquid of interest. A poor glow piece is then used to illuminate these subdivisions. Using a camera, dispersed light was then stored in subsequent image outlines with realised stretches. PIV evaluates the entire speed field and calculates the atom's elimination within the set time period by capturing two photos, one after the other, with a rapid camera (Fujita et al., 1998). These images are then processed on a computer using Matlab routines to analyse the atom development in subsections of the PIV images using cross relationship techniques. After considering the picture amplification and temporal delay, the result inspires a molecular picture uprooting design.

Maynord (1985) depicted the upstream content of the spillways in four different states: one vertical and three tilted. The downstream bend profile is the bit between the peak pivot and the digression segment, while the upstream bend profile is a mix of radii that are comparable to the whole head.

(2001, Savage and Johnson) The spillway's rating bend is a crucial pressure-driven hallmark that demonstrates the model's constancy and precision. It is calculated based on hypothetical circumstances. An electromagnetic flowmeter is used to record the data, which is then compared to the condition. The flowmeter was installed beneath the flume. The constructed bay tank is connected to the radial syphon. The possible spillway release through the ogee-peaked spillway can be depicted.

(Falvey, 1990). When cavitation occurs close to a spillway's significant surface, it causes the formation of fume bubbles. The air pockets dissolve in the water and are carried along by the stream. As a result, the air pockets' encircling pressuring factor grows, rendering them useless at this point and causing them to collapse. The collapses occur often and with an incredibly high pushing factor of up to 1,500 MPa.

(Lesleighter, 1988) and sway the solid surface inexorably. This cycle results in the weary disappointment of the significant materials, which causes microcracks on the surface. These undesirable fractures will eventually result in a prolonged opening. The aperture widens with time, with the swift stream impacting the downstream end.

Sheehan (1974) conducted trials on large materials and discovered that an air content of 1–2% significantly reduced cavitation damages, whereas an air content of more than 5–7% caused no problems.

Chen et al. (2003), referenced from Perceptions of models cavitation damages were significantly reduced when the air focus at the divider was between 1.5 and 2.5 percent. When the fixation reached 7–8%, the risks eliminated completely. An aerator is a clever device that artificially adds air to the water stream in a spillway. It's placed where cavitation damage might occur. The Grand Coulee Outlet Works demonstrated adequate air circulation in 1961.

(Borden et al., 1971). Aerators have become virtually limitless since then. In a chute spillway, an aerator helps to raise the air grouping of the water stream. A free stream is created as the water passes through the aerator, and furious swirls in the lower fly surface successfully entrain air. Under the stream, an air pit is formed. The pressing factor in the cavity falls below the climatic pressing factor when the fly entrains air. As a result, there is a critical factor differential between the cavity and the environment, allowing air from the air to be sucked into the hole via the aerator framework.

Bercovitz et al. (2016) used a large-scale PIV to study the surface speed field of a plummeting water fly from a sharp-peaked weir. They wanted to think about the nappe orientation and signature duration, as well as the energy distribution.

Lin et al. (2008) used the BIV and HSPIV approaches to get air circulation district speed fields in a stream at a drop structure. The inventors demonstrated the application of these predictions and methodologies to an aerator stream in a flume in Papers VII. Hibiki and Ishii, 2000; Kolev, 2005; ANSYS, 2015; Zkan et al., 2016). The Volume-of-Fluid (VOF) Model, Mixture Model, and Two-Fluid Model were among them (TFM). Aerator streams are air-bubble streams that occur at a high stream speed. It puts mathematical predictions to the test, as stream speeds routinely exceed 20 m/s and can reach 45 m/s in massive dams. The increased speed increases commerce between air and water, increasing the importance of air transportation. This implies that in an aerator stream, the stage connections' intentions get muddled. The absence of model estimation information for mathematical model alignment and check is the primary rationale for this.

(Kökpnar and Göüş, 2002; Deng et al., 2005; Liu and Yang, 2014; Jothiprakash et al., 2015; Rahimzadeh et al., 2015). The Mixture Model was used by Aydin and Ozturk (2009) and Zhang et al. (2011) to investigate an aerator stream. They discovered that the model had the best capability for reproducing aerator streams, especially in areas with strong air-fixation. These investigations looked at both test and model data and found that the model took into account a reasonable base pressing factor and air hole patterns.

(Zhang, 2008; Xu et al., 2001). The Two-Fluid Model is also an optional model for illustrating aerator streams. It illustrates the partnerships between air pockets and water, including the stage drag power, the violent scattering power, and so on, unlike the VOF model. In similar studies, the mathematical outputs are compared to the exploratory data. These tests demonstrate the capacity to exhibit aerator streams and elicit further evaluations and analyses of the Two Fluid Model.

Aydin et al. (2019) Using a CFD model, investigated the base exit of the Ilsu Dam. The displaying of air circulation displays with various plans was investigated throughout the examination. Finally, two novel air circulation designs for that particular situation were developed, despite recent breakthroughs.

Chanson (2009) highlighted concern about extrapolating lab results to large model pressure-driven structures, and explored the unique similarity of air entrainment metrics. Actual model investigations were conducted for the most part using the Froude similitude rule with a lower Reynolds number than comparing model flow, and the concept of scale effect is closely linked to the assessment of major trademark air–water flow attributes, according to the report.

Kumcu (2017) investigated the flow through a full sized (model) ogee spillway using a CFD model using Flow-3D and compared the results to 1/50 scaled actual model results. In terms of free-surface attributes like surface level, flow speed, and pressing factor, the mathematical model's aftereffects closely match the scaled real model; however, no information on the air entrainment sum and its scale effects was provided, save from some air fixation.

Geun and Hyun (2005) investigated some of the ogee spillway's characteristics using a CFD model (Flow-3D) to detect discomfort and scale effects. They discovered that whereas surface unpleasantness and scale effects have a minor impact on a few outcomes such as release, water surface, and peak pressures, they have a significant impact in the largest speed region. Ferrari (2010) produced an excellent mathematical report on the free surface flow over a sharp-peaked weir. By contrasting the free-surface profiles obtained from trial estimations in the writing, the results were confirmed, and a good comprehension was achieved.

Heller (2011) discussed how scale effects were avoided or modified using the Froude and Reynold model-model similitudes to depict scale effects for conventional pressure driven flows. Felder and Chanson (2017) investigated scale effects in air–water flow using high-speed blending flow. They conducted a thorough investigation of the air–water flow features, such as the interfacial area, choppiness qualities, and molecule sizes, all of which might be influenced by scale effects. They also stated that the examination's findings are applicable to the next air–water flow type, but that model information was necessary for final confirmation.

While scale effects are minor for various flow limits, such as release flow rate, water surface, and pressing factors, it tends to be highly significant for high-speed air–water mix flow, such as spillway flows with an aerator, according to the aforementioned experts. The model for this study was a spillway aerator from an RRC type dam with a height of 100 metres. The spillway's mathematical model was created in the model using a computational fluid model under various flow circumstances (5223, 3500, 1750, and 1000 m3/s of flow rate), and the existing plan's hydrodynamic conduct was analysed using computational fluid elements (CFD). The obtained mathematical model results were compared to the aftereffects of the DSI model test, and the collected results were assessed using the present plan's hydrodynamic characteristics (zcan 2011). A single stage flow model was used to investigate the hydrodynamics of the flow on the spillway, and a double phase (air–water) flow model was used to execute the aerator on the spillway.

Chadwick et al. (2004) depicted a spillway, according to is a meticulously prepared entrance used to handle the controlled arrival of water from a dam into a downstream location, usually the canal that was dammed. Spillways safely drain floodwaters so that the water does not rise over the design, causing damage or even failure of the dam.

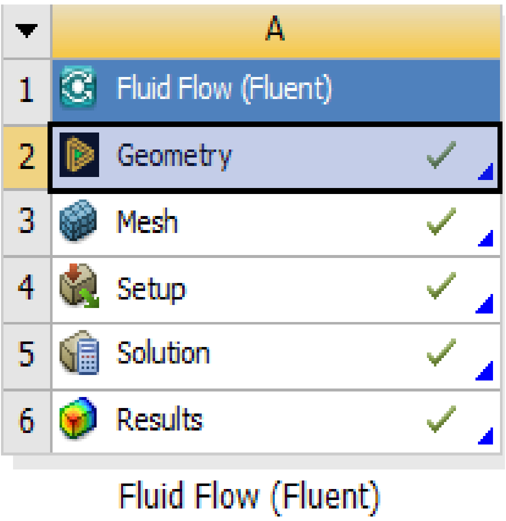
(Van Vuuren and colleagues, 2011) There is a wealth of information available on estimating the Ogee profile for spillway design, and a few projects have been undertaken to develop a relationship that would allow the condition of the Ogee bend to be mathematically shown using 2-dimensional stream boundaries. Regrettably, the stream over an Ogee spillway can't always be thought of as a 2-dimensional stream condition. The asymmetry of valleys and the geographical methodological channels where spillways are built will have an influence on the stream example and speed up circulation upstream of the spillway. If these 3-dimensional stream practices are ignored, the Ogee spillway construction might be compromised. The most distinct 3-dimensional stream boundaries that influence the Ogee profile computation are included.

CHAPTER THREE

METHODOLOGY

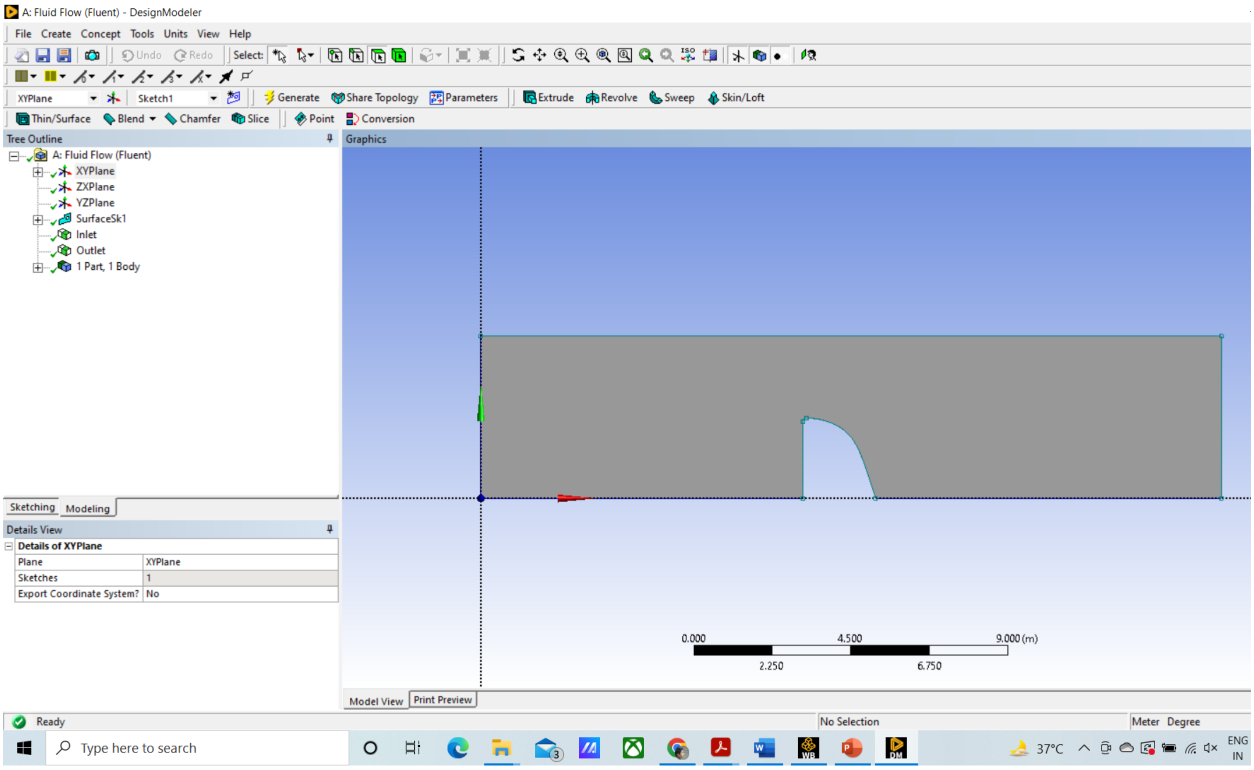
Ansys Fluent consists of 5 stages :

1) Geometry
2) Mesh
3) Setup
4) Solution
5) Results



3.1) Geometry

Using the Ansys Design Modeller, CFD Model Geometry was built.



After creating the sketch, go to Concept menu→ Select surface from sketch option→ Select the sketch drawn→ Then click generate.

Select the inlet part, Right click on modelling space→ Select named selection→ Rename as Inlet→ Then click generate.

Select the outlet part, Right click on modelling space→ Select named selection→ Rename as Outlet→ Then click generate.

Geometry Dimensions :

Length of Flume : 21.189 m

Height of Flume : 4.6378 m

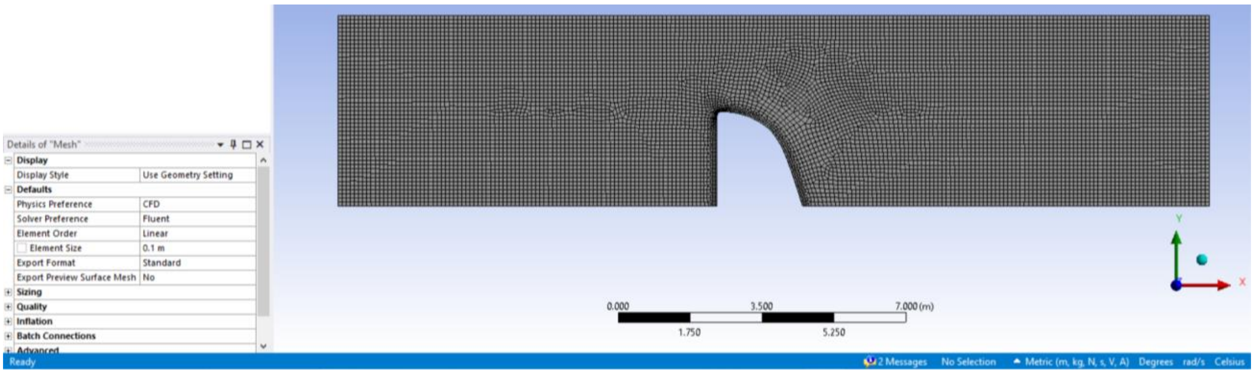
Height of Spillway : 2.2003 m

Length of Spillway : 2.0789 m

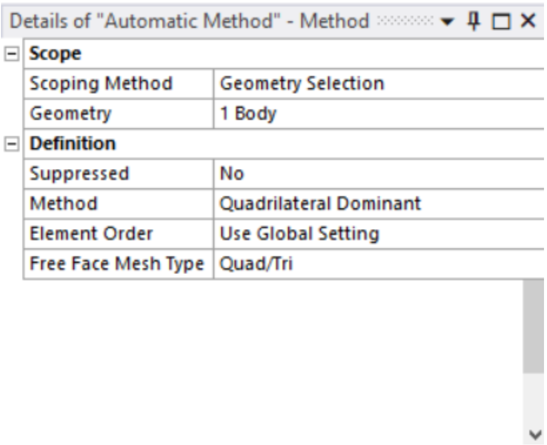
Head over Spillway : 2.3375 m

3.2) Mesh

The meshing process is a very important stage in CFD modelling which requires much attention. In order to analyse the fluid flow, the domain had to be split into smaller cells within which the governing equations would be solved.



Element Size : 0.1 m



Details of "Automatic Method" - Method

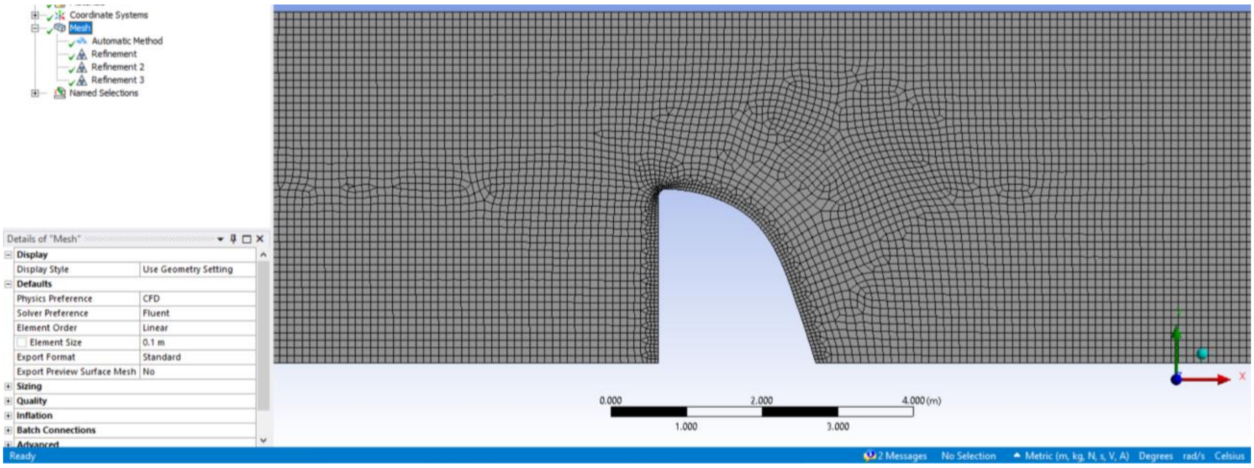
Scope	
Scoping Method	Geometry Selection
Geometry	1 Body
Definition	
Suppressed	No
Method	Quadrilateral Dominant
Element Order	Use Global Setting
Free Face Mesh Type	Quad/Tri

Quadrilateral Dominant type of meshing was selected.

The main features of interest in the numerical domain are:

- the spillway crest,
- the downstream part where the turbulent phenomenon occurs, and
- the region of interface between the two phases (water and air).

Therefore Refinement needs to be applied in these regions.



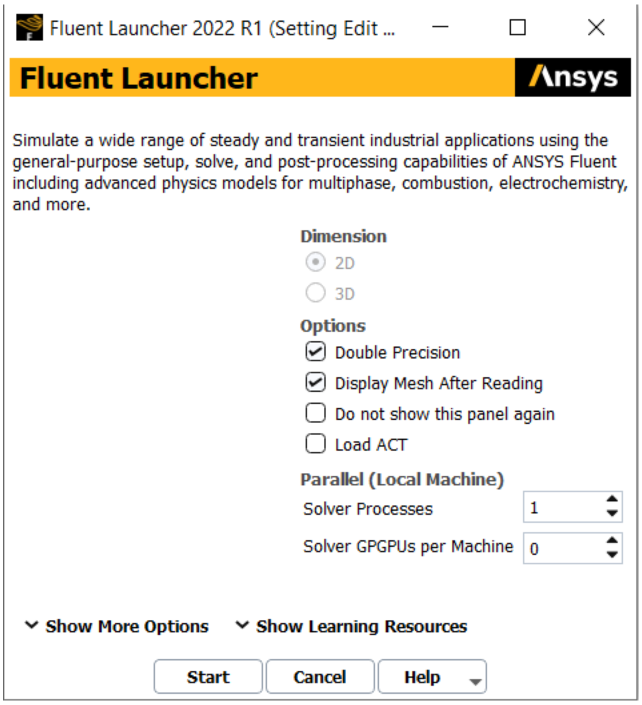
Refinement applied as shown in figure.

Another important indicator of mesh quality that ANSYS FLUENT uses is a quantity referred to as the orthogonal quality. The orthogonal quality (OQ) is derived directly from Fluent solver discretization. The range of orthogonal quality is 0-1, where the minimum orthogonal quality must be greater than 0.051.

3.3) Setup

After creating the geometry of the model and meshing it up in the Ansys-Fluent package, the mesh was imported into fluent solver. Various parameters within the solver had to be set before the simulations could be started.

Tick Double Precision before launch of setup.



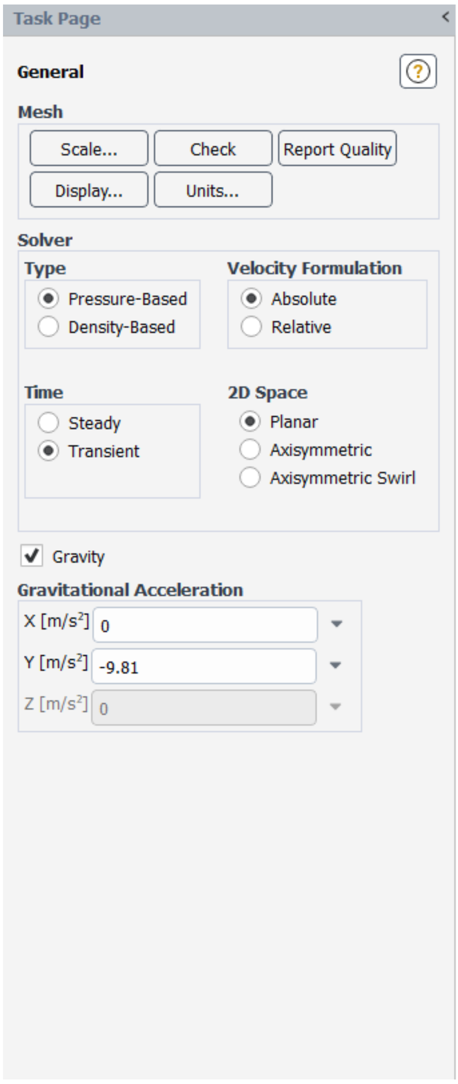
The general settings are first defined.

Type→ Pressure Based

Velocity Formulation→ Absolute

Time→ Transient

Gravity is enabled & defined as Y→ -9.81 m/s^2

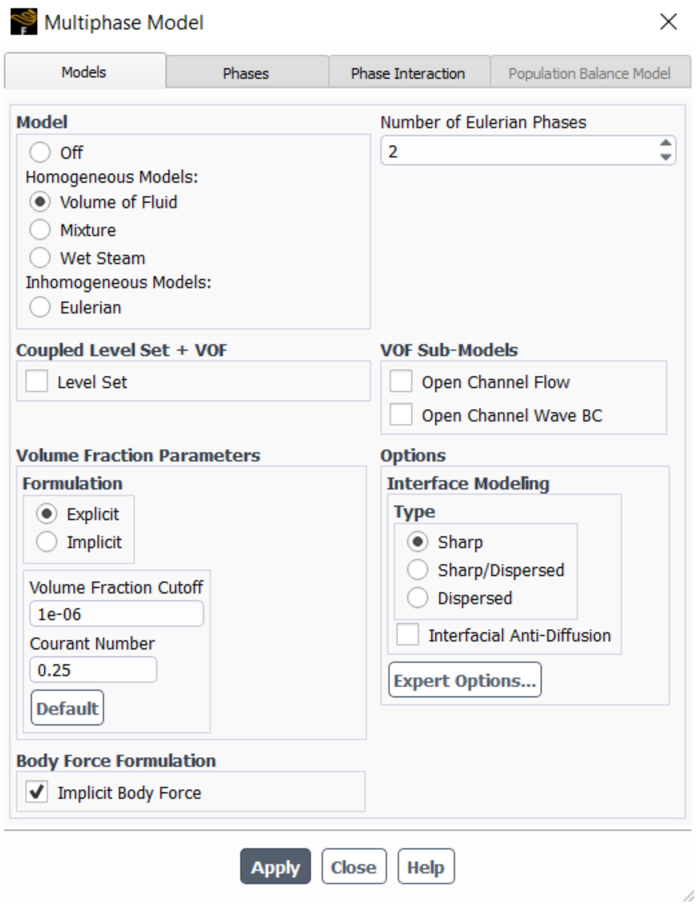


3.4) Multiphase model selection

The volume of fluid model (VOF) was chosen to simulate the multiphase flow.

Two eulerian phases: air and water, were defined as primary and secondary phase respectively.

Another alternative for setting eulerian phases was tried, that was, to define water as primary phase and air as secondary phase but this resulted into flow instability.



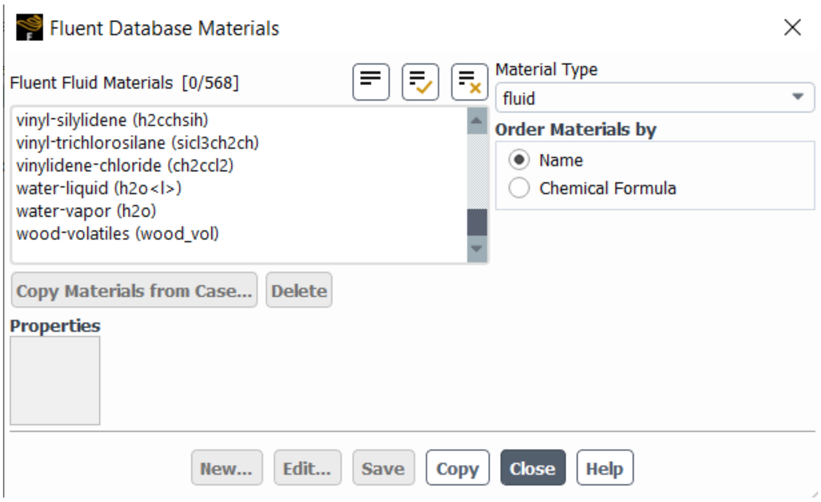
Implicit Body Force option is ticked.

Formulation→ Explicit

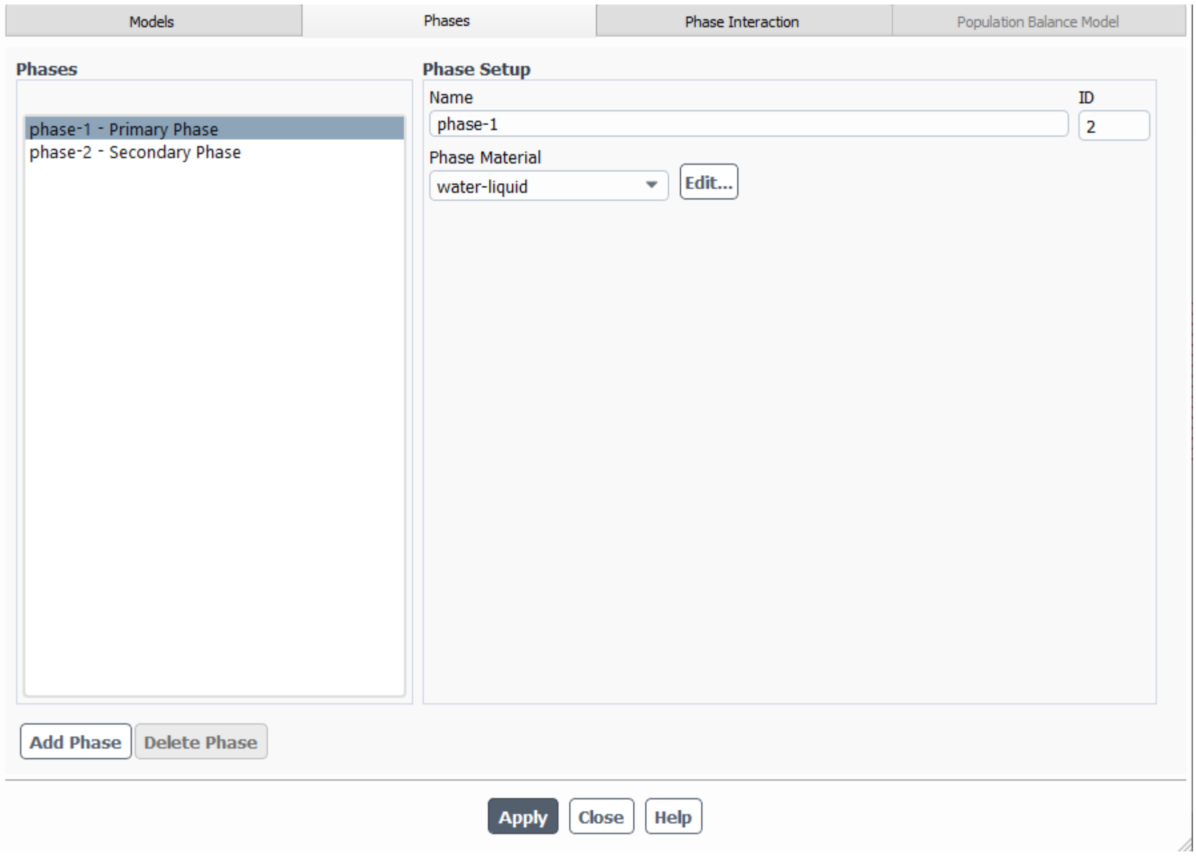
Number of Eulerian Phases→ 2

Volume Fraction Cutoff→ 1e-06

Courant Number→ 0.25

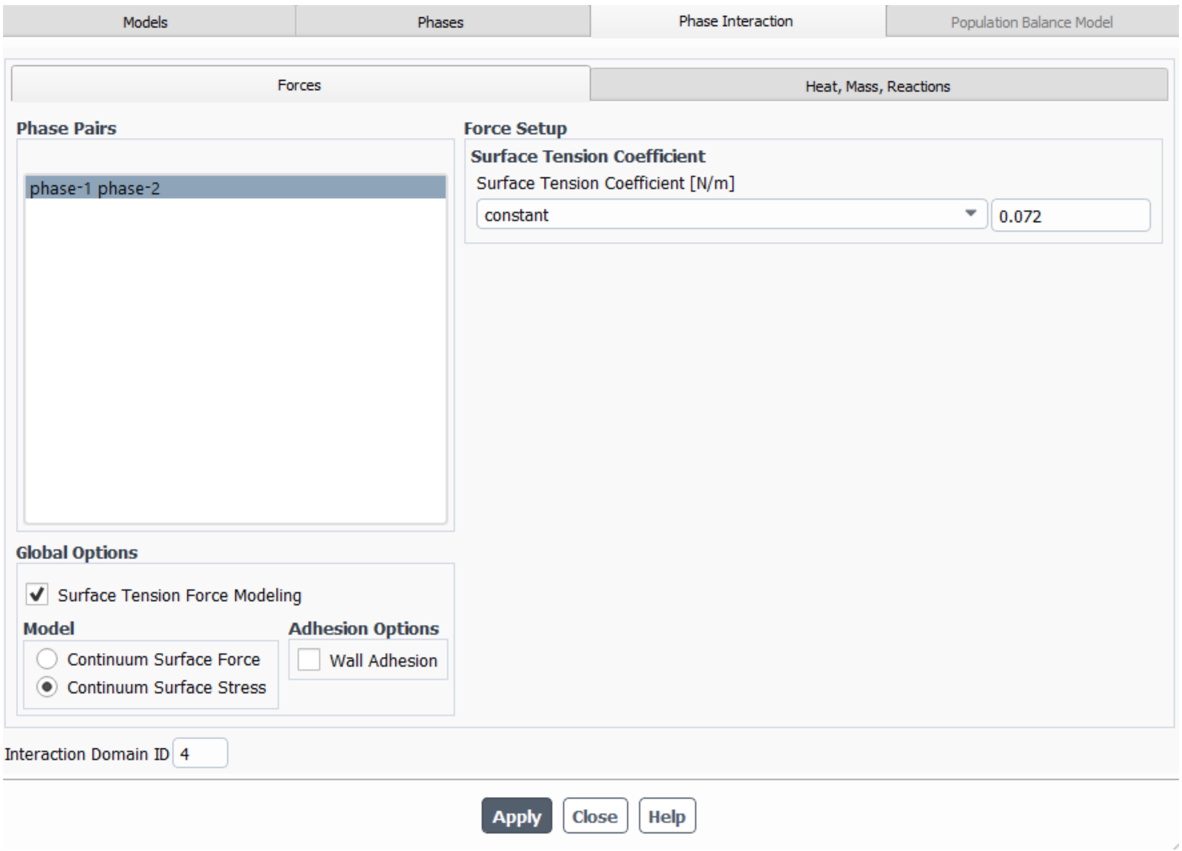


From Fluent Database Materials, Water is copied.



Primary Phase→ Water

Secondary Phase→ Air

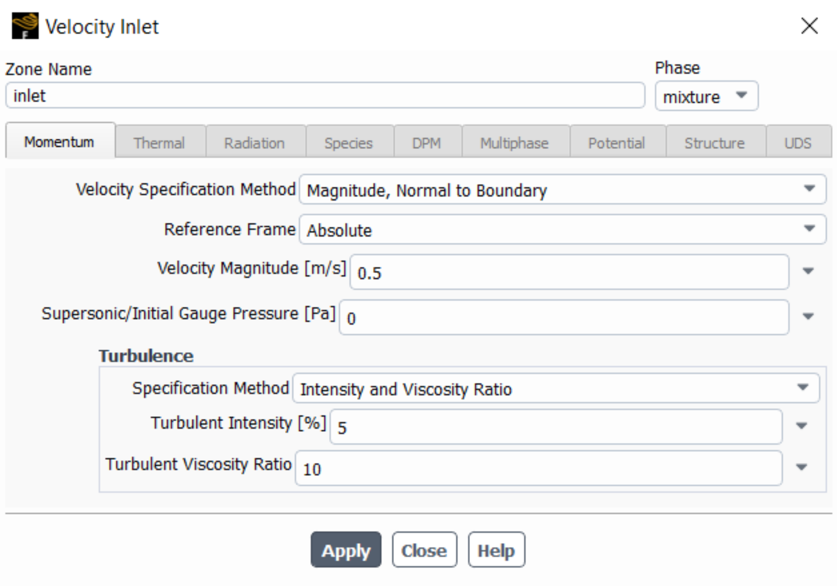


Model→ Continuum Surface Stress Model

Surface Tension Coefficient→ 0.072

3.5) Boundary conditions

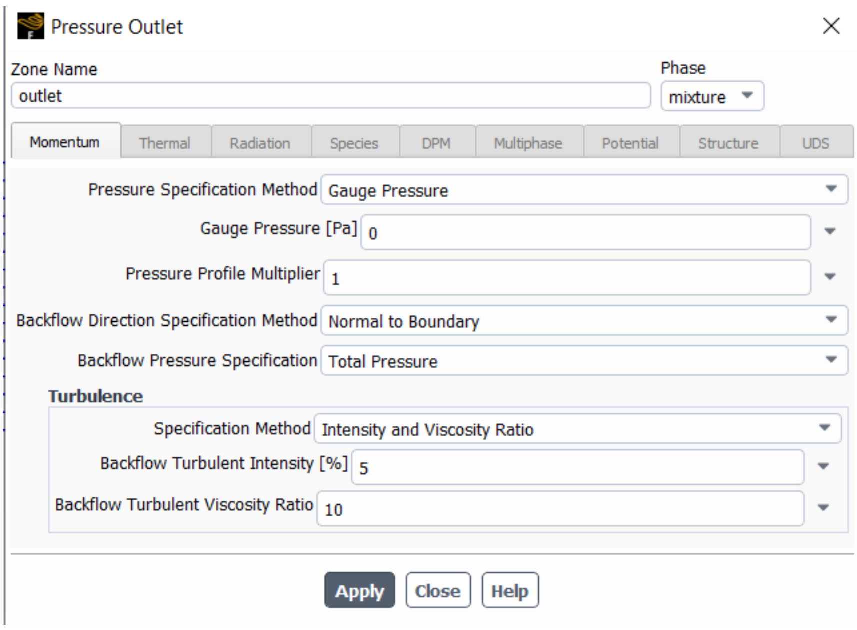
The precision of boundary conditions on the solution domain plays a capital role for the accuracy of the results. The boundary conditions to be specified consist of flow inlet and outlet boundaries, which have to be defined with the flow properties such as turbulence parameters, velocity and pressure.



Velocity Magnitude→ 0.5 m/s

Turbulent Intensity→ 5%

Turbulent Viscosity Ratio→ 10

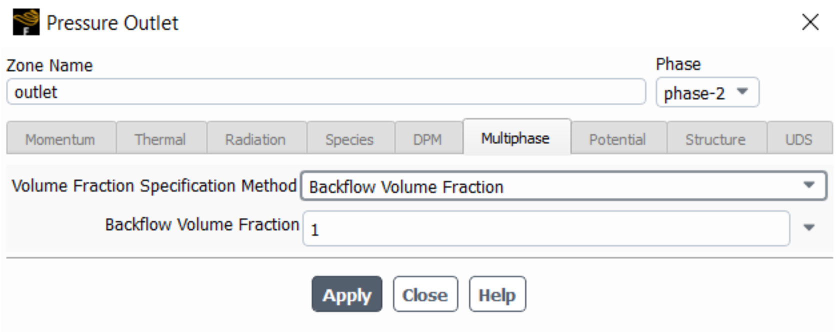


Backflow Direction Specification Method→ Normal to Boundary

Backflow Pressure Specification→ Total Pressure

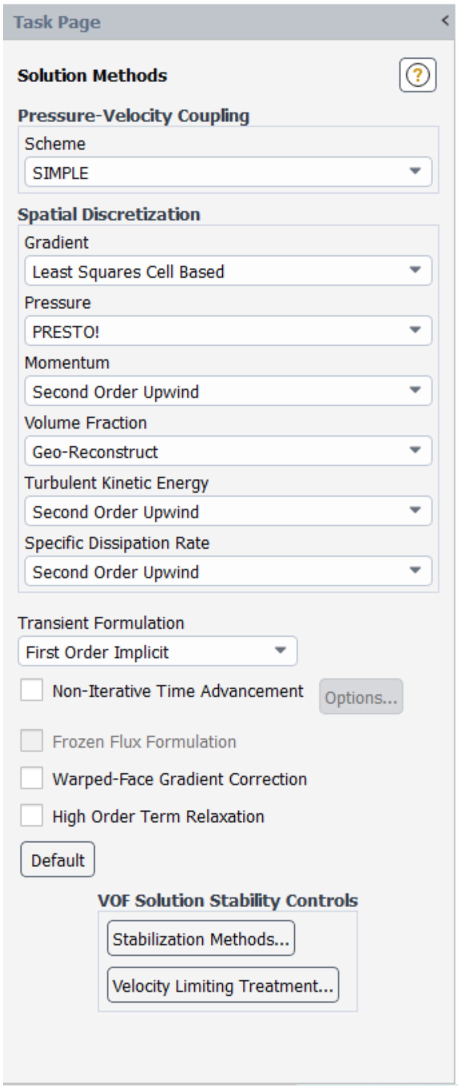
Backflow Turbulent Intensity→ 5%

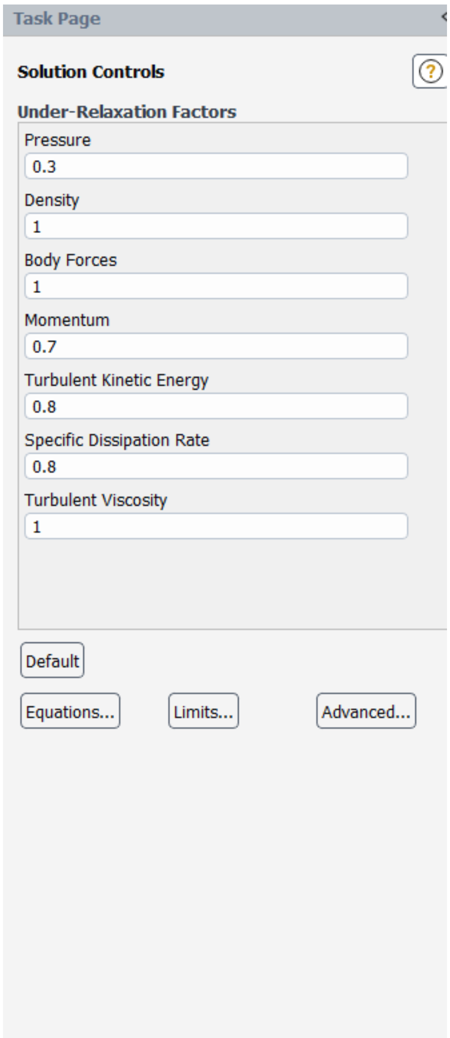
Backflow Turbulent Viscosity Ratio→ 10



In Pressure Outlet→ Select Phase 2→ Backflow Volume Fraction→ 1

3.6) Solution





The under-relaxation factors were used in the pressure-based solver to stabilize the convergence behaviour.

Under Relaxation Factors are defined as :

Pressure→ 0.3

Density→ 1

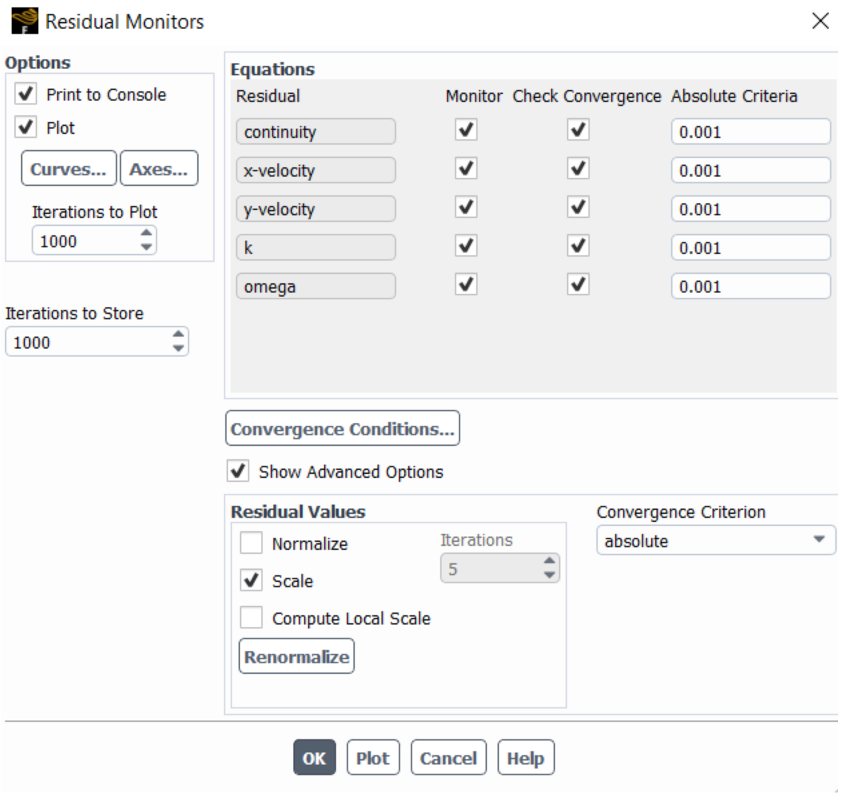
Body Forces→ 1

Momentum→ 0.7

Turbulent Kinetic Energy→ 0.8

Specific Dissipation Rate→ 0.8

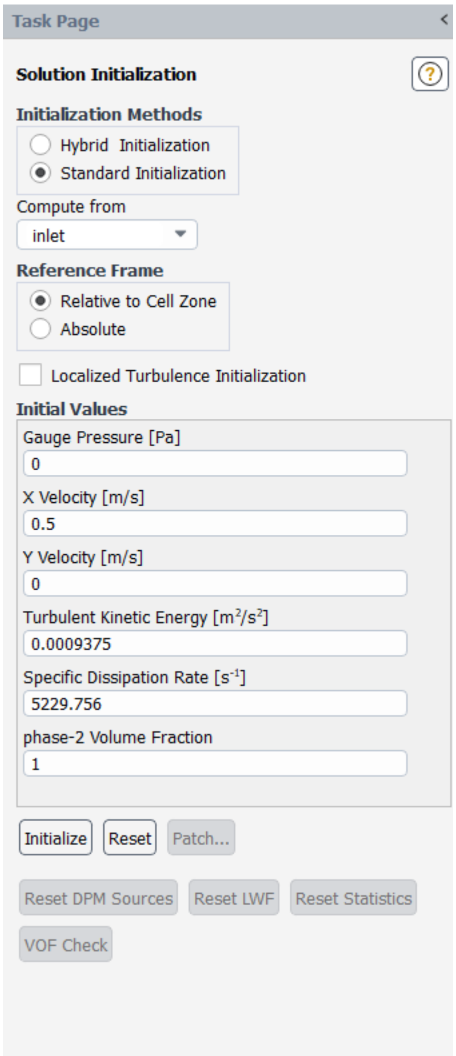
Turbulent Viscosity→ 1



Convergence Criterion→ Absolute

3.7) Initialization

For each test, it is necessary to initialise the model. This is done by setting initially all pressures to the atmospheric conditions. For these conditions, the solution domain was entirely filled with air. The flow computations were set to start from the inlet boundaries to allow stable conditions to be far from the spillway crest.



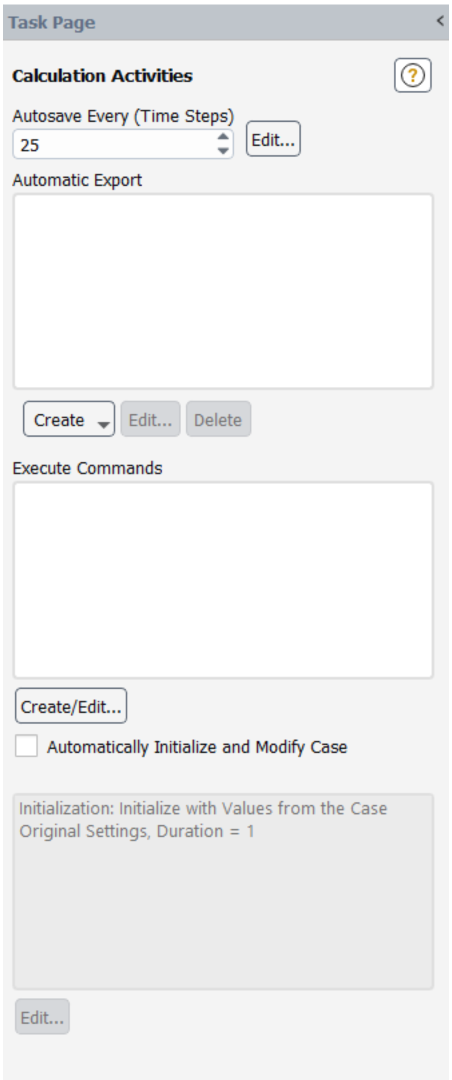
Initialization Method→ Standard Initialization

Compute from→ Inlet

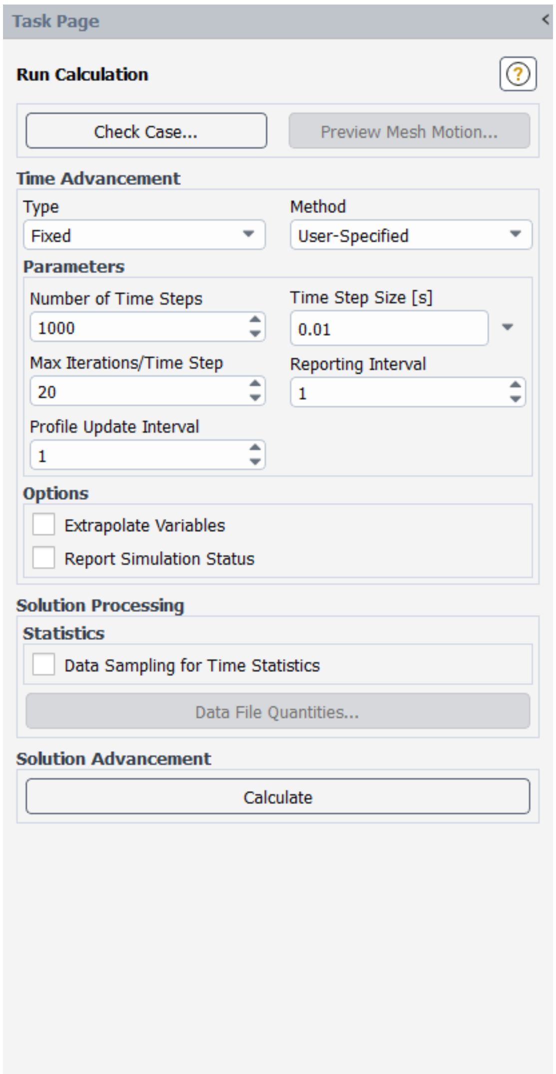
X Velocity→ 0.5 m/s

Phase 2 Volume Fraction→ 1

3.8) Calculation Activities



Autosave every→ 25 Time steps



Parameters for calculation are :

Number of Time Steps→ 1000

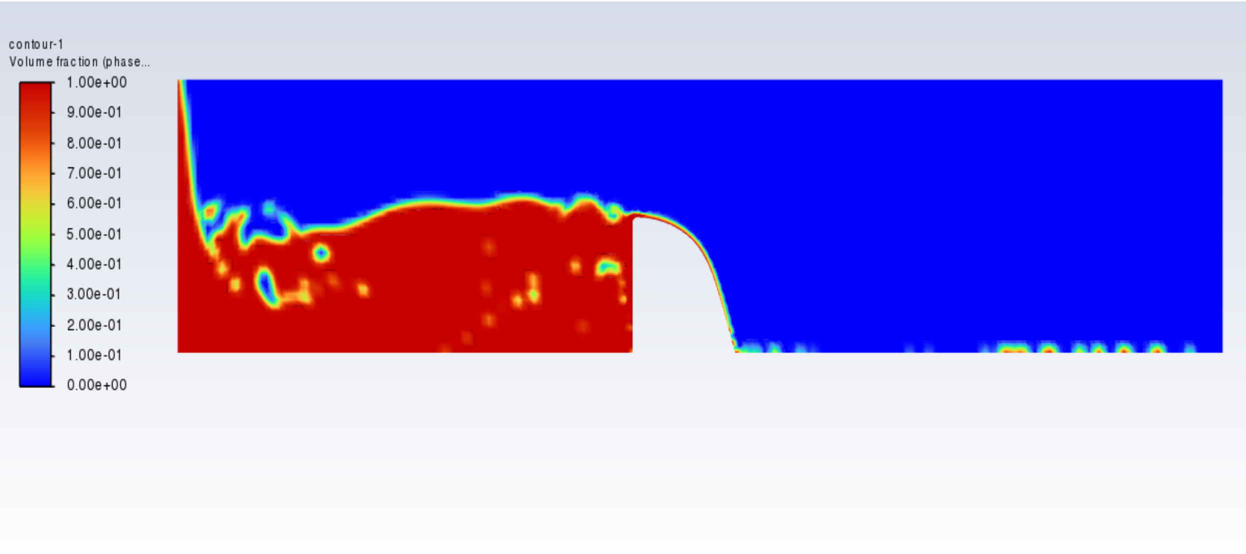
Time Step Size→ 0.01 s

Max Iterations/Time Step→ 20

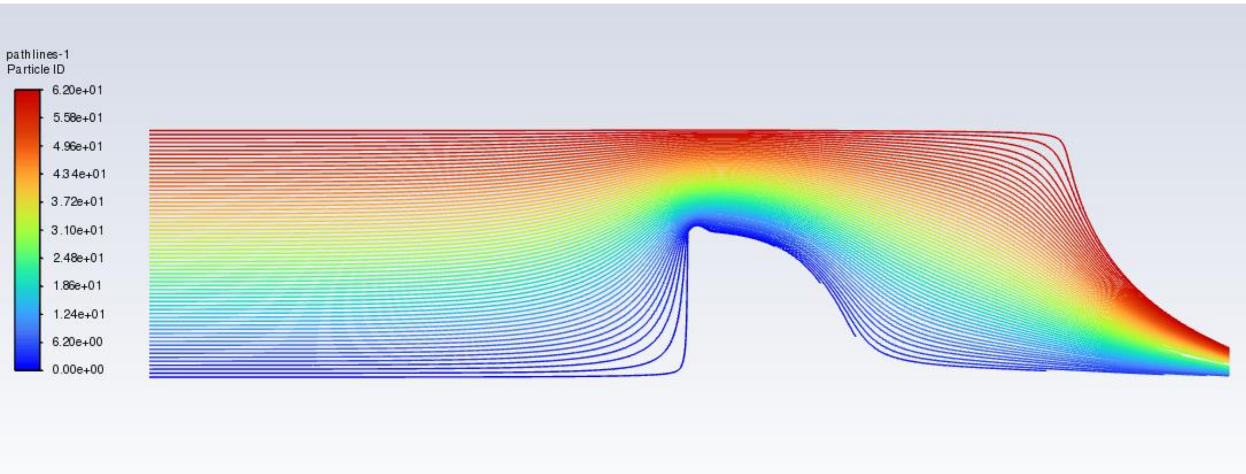
3.9) Flow development

With the large extent of data recorded during simulations, different ways were used to visualise flow features. The density and pathline contours were employed in the visualisation of flow pattern in the domain.

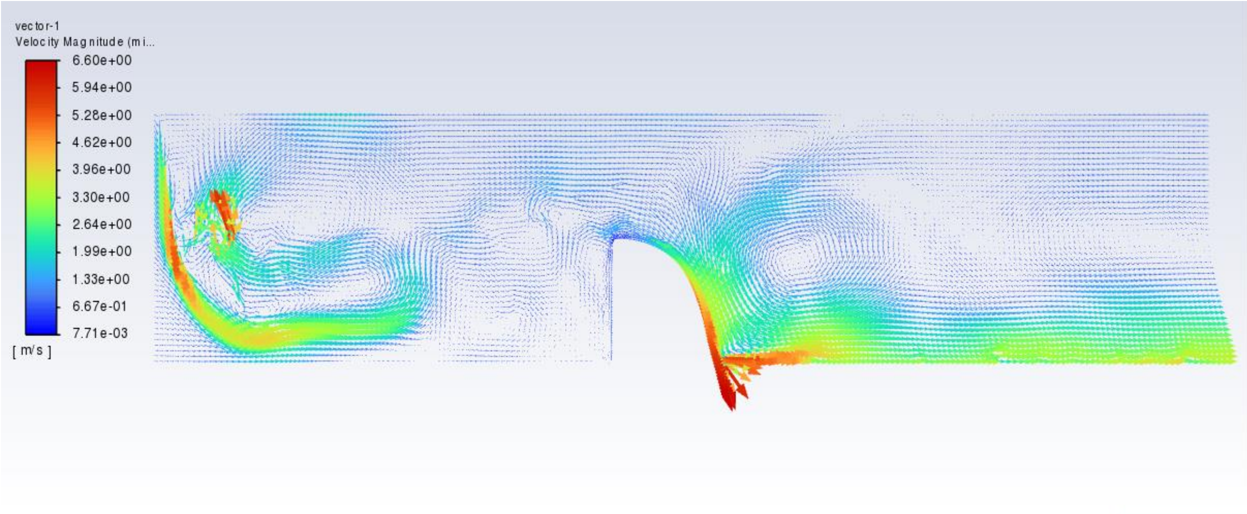
Contours of Volume Fraction



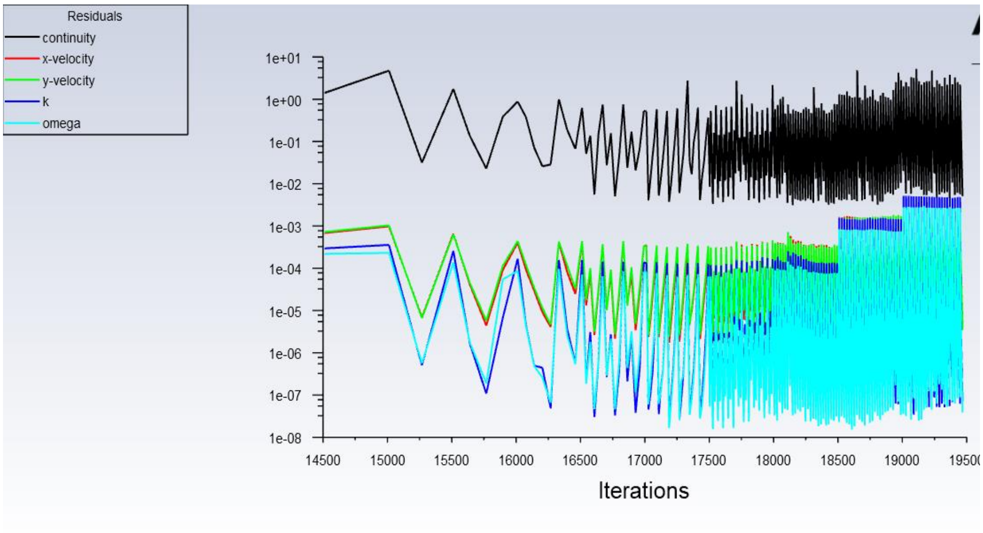
Simulated Pathlines



Velocity Vectors



Scaled Residuals (Iterations)

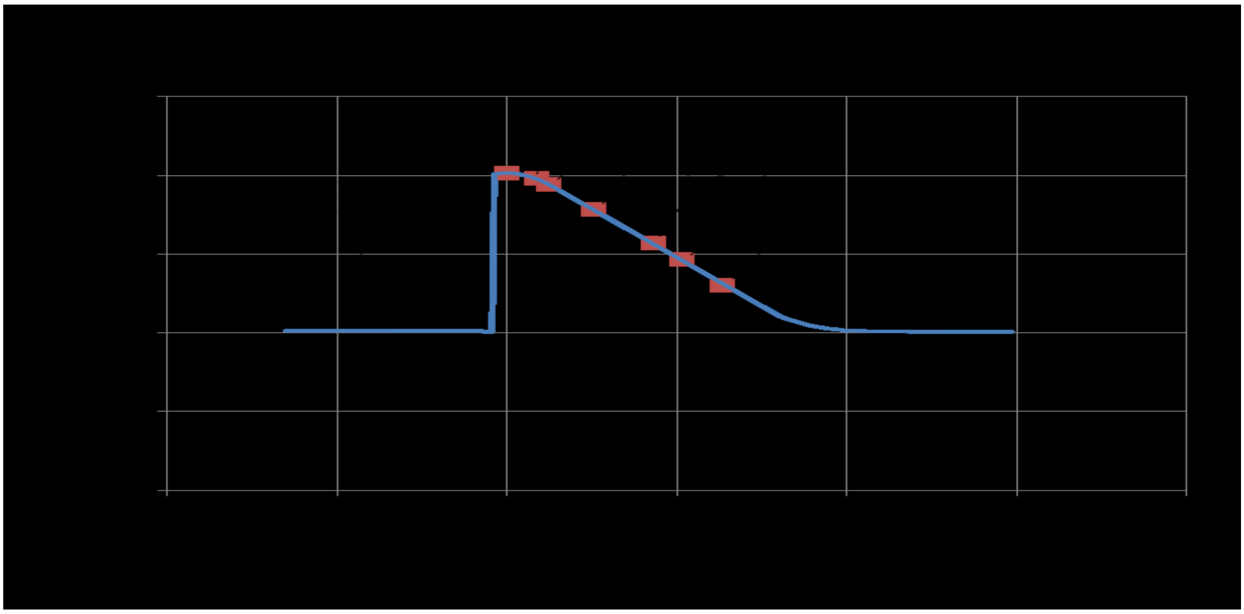


3.10) Pressure Results

Flow rate (l/s)	Sensor 1	Sensor 2	Sensor 3	Sensor 4	Sensor 5	Sensor 6	Sensor 7
	CFD modelling: Average Sensor pressure (m)						
24	0.020	0.007	0.014	0.004	-0.007	-0.029	0.001
36	0.015	0.006	0.017	0.008	-0.005	-0.033	0.004
42	0.010	0.005	0.019	0.010	-0.004	-0.035	0.006
57	-0.004	0.001	0.020	0.015	0.001	-0.036	0.011

72	-0.024	-0.006	0.019	0.021	-0.004	-0.037	0.016
90	-0.050	-0.014	0.017	0.027	0.006	-0.035	0.021
109	-0.082	-0.023	0.013	0.034	0.010	-0.033	0.028
131	-0.122	-0.036	0.007	0.040	0.015	-0.029	0.035

Location of Pressure Sensors



CHAPTER FOUR

CONCLUSIONS

1) The pressure results obtained from physical and CFD modelling show similar pressure distribution patterns on the spillway face and the pressures are in reasonable agreement.
2) However, CFD models did not simulate the pressure fluctuations as observed in the physical model pressures.
3) Mathematical modelling plays a capital role in the design and analysis of hydraulic structures. However, this often necessitates physical models, considered as the most established form of hydraulic modelling, to ensure reasonable accuracy of the results. This study was devoted to carrying out a CFD validation of ogee spillway hydraulics.
4) The pressure readings obtained, a negative pressure reading was encountered for the discharges greater than the design discharges with a clear flow separation from the crest. In

addition, an accentuated sub-atmospheric pressure appeared, especially for heads which are greater than 1.33Ho.

5) For CFD modelling, the model domain was developed with the same dimensions as the physical model with regard to minimising errors as much as possible. Quadrilateral Dominant mesh in 2D was selected where the minimum grid size in 2D models was 0.1m. The trend towards mesh independence was observed by comparing the pressure reading obtained for different grid sizes.
6) The Volume of Fluid (VOF) and the Realisable "$k-\varepsilon$" models chosen for this study to model the multiphase flow and turbulence, respectively, simulated successfully the flow over an ogee spillway. The investigations proved that CFD models were able to develop a flow surcharge and simulate the pressures similar to the physical model test results. However, they were unable to accurately determine the pressure fluctuations similar to those obtained from physical modelling.
7) Concerning the comparison between physical and CFD modelling, it is clear that physical modelling still proves to be the more established of the two methods. Although CFD tools still have limitations (including grid resolution, run times and numerical instabilities to name a few), there are many instances where they may offer an increased accuracy over the designs and provide an insight into the required application. CFD models can provide more detail about velocity and turbulence than a physical model can and may be more economical in some cases.

CHAPTER FIVE
RECOMMENDATIONS

1) This study attempted to validate the uncontrolled ogee spillway hydraulics with Ansys-Fluent software. The same validation is recommended for a controlled ogee spillway to assess the use of this software in the design and testing processes of this particular type.
2) Other types of turbulence models, apart from Realisable $k-\varepsilon$ and Volume of Fluid model (VOF) within Ansys-Fluent should be assessed to determine their capabilities for turbulence and multiphase modelling.
3) Ansys-Fluent takes a very long time to produce results. Normally, a five minutes test length in physical modelling would take more than one month to provide results for a single

discharge scenario. Therefore, methods should be developed to reduce simulation periods e.g. By means of computer clusters.

REFERENCES

[1] Arstid, P. : Reduction of Process Simulation models: A Proper Orthogonal Approach, Ph.D. thesis, Technische Universiteit Eindhoven (2004)

[2] Ito, K., Ravindran, S. S., A Reduced-Order Method for Simulation and Control of Fluid Flows, J. of Comutational Physics 143, p. 403–425 (1998)

[3] Rousset, M., Reduced Order Modeling for Thermal Simulation, Master's Thesis, Stanford University, 2010.

[4] Brenner, T. A., et. al., A reduced-order model for heat transfer in multiphase flow and practical aspects of the proper orthogonal decomposition, Computer and Chemical Engineering 43 (2012) p.68-80.

[5] Lappo, V., Habashi, W., Reduced Order POD/Kriging Modeling for Real Time 3D CFD, 11th Pan-American Congress of Applied Mechanics, January 04-08, 2010, Brazil.

[6] Lieu, T., et. al., Reduced Order fluid/structure modeling of a complete aircraft configuration, Comput. Methods Appl. Mech. Engrg, 195 (2006), p.5730-5742.

[7] Francis H. Harlow, J. Eddie Welch, Numerical Calculation of Time Dependent Viscous Incompressible Flow of Fluid with Free Surface, The Physics of Fluids, Volume 8, Number 12, December 1965.

[8] Tome, M. F., McKee, S., GENSMAC: A Computational Marker and Cell Method for Free Surface Flows in General Domains, J. of Computational Physics 110, p.171-186 (1994)

[9] Girault, V., A Combined Finite Element and Marker and Cell Method for Solving Navier-Stokes Equations, Numerical Mathematics. 26, p.39-59, (1976).

[10] Saleh, H., Hashim, I., Conjugate Heat Transfer in Rayleigh-Benard Convection in a Square Enclosuse, The Scientific World Journal, Vol. 214, Article ID: 786102

[11] McKee, S., Review The MAC Method, Computer & Fluids 37 (2008) p.907-930.

[12]McKee, S., et. al., Recent Advances in the Marker and Cell Method, Arch. Comput. Mech. Engng., vol.11, 2, p.107-142 (2004).

[13] M. B. Liu, G. R. Liu, Smoothed Particle Hydrodynamics (SPH): an Overview and Recent Developments, Archives of Computational Methods in Engineering, Volume 17, Issue 1, pp 25-76, March 2010.

[14] Szewc , K., A Study on Application of Smoothed Particle Hydrodynamics to Multi-Phase Flows, Int. J. Nonlinear Sci. Numer. Simul. 2012; 13(6): p. 383–395

[15] Bockmann, A., et. al. Incompressible SPH for free surface flows, Computers & Fluids 67 (2012) p.138̦151

[16]Valizadeh, A., et. al. Modeling Two-Phase Flows Using SPH Method, J. of Applied Sciences 8(21):p.3817-3826, (2008).

[17] Jeong, J.H., et. al., Smoothed particle hydrodynamics:Applications to heat conduction, Computer Physics Communications 153 (2003) p.71–84

[18] Rook, R., Modeling Transient Heat Transfer Using SPH and Implicit Time Integration, Intl. J. of Computation and Methodology, 51:1, 1-23 (2007)

[19] Szewc, K., Pozorski, J., Multiphase heat transfer modelling using the Smoothed Particle Hydrodynamics method, Computer Method in Mechanics - 2013, Poland.

[20] Krog, O. E., Elster, A. C., Fast GPU-based Fluid Simulations Using SPH, Para 2010 – State of the Art in Scientific and Parallel Computing – extended abstract no. 139, University of Iceland, Reykjavik, June 6–9 2010

[21] Auer, S., Real-time particle-based fluid simulation, Master Thesis, Technische Universität München, Germany.

[22] L. Greengard and V. Rokhlin, A Fast Algorithm for Particle Simulations, J. of Computational Physics 73, 325-348 (1987)

[23] H. Cheng, et. al.,A Fast Adaptive Multipole Algorithm in Three Dimensions, J. of Computational Physics 155, 468–498 (1999)

[24] Greengard, L., Kropinski, M. C., An Integral Equation Approach to the Incompressible Navier-Stokes Equations in Two Dimensions, SIAM J. Sci. Comput. Vol. 20, No. 1, p. 318-336 (1998).

[25] Yokota, R., et. al., Petascale turbulence simulation using a highly parallel fast multipole method on GPUs, Computer Physics Communications 184 (2013) p. 445–455.

[26] V.D. Kupradze, M.A. Aleksidze, The method of functional equations for the approximate solution of certain boundary value problems, USSR Comput. Math. Phys. 4 (1964) p.82–126.

[27] Chen, C. S., et. al., The Method of Fundamental Solutions – A Meshless Method, Dynamic Publishers, Inc. Atlanta, 2008.

[28] Zuosheng, Y., The Fundamental Solution Method for Incompressible Navier-Stokes Equations, Int. J. Numer. Meth. Fluids 28: p.565–568 (1998)

[29]Young, D. L., et. al., The method of fundamental solutions for solving incompressible Navier–Stokes problems, Engineering Analysis with Boundary Elements 33 (2009) p.1031–1044.

[30] Gaspar, C., Solving Interface Problems by the Regularized Method of Fundamental Solutions, 6th European Conf. on CFD, July 2014, Spain.

[31] Xiong, X. T., A Numerical method for Identifying Heat Transfer Coefficient, Appl. Math. Modelling 34 (2010) p.1930˛1938

[32] Valle, M. F., et. al., Estimation of the heat transfer coefficient by means of the method of fundamental solutions, Inverse Problems in Science and Engineering Vol. 16, No. 6, (2008), p.777–795.

[33] Kuhnert, J., Tiwari, S., Finite pointset method based on the projection method for simulations of the incompressible Navier Stokes equations, Fraunhofer Institute for Industrial Mathematics, Kaiserslautern, Germany.

[34] Tiwari S., Kuhnert J., Modeling of two-phase flows with surface tension by finite pointset method (FPM), Journal of Computational and Applied Mathematics 203 (2007) p.376 – 386

[35] Tiwari, S., et. al.,A Meshfree Method for Simulations of Interactions between Fluids and Flexible Structures, Fraunhofer Institute for Industrial Mathematics, Kaiserslautern, Germany.

[36] Resendiz-Flores, E.O., Garcia-Calvillo, I. D., Application of the finite pointset method to non-stationary heat conduction problems, International Journal of Heat and Mass Transfer 71 (2014) p.720˛723.

[37] Tokura, S., Comparison of Particle Methods : SPH and MPS, 13th International LS-DYNA User Conference.

[38] Ataie-Ashtiani, B., Farhadi, L., A Stable moving-particle Semi Implicit Method for Free Surface Flows, Fluid Dynamics Research 38 (2006), p.241-246.

[39] Kondo, M., et. al., Incompressible Free Surface Flow Analysis Using Moving Particle Semi-Implicit Method, Joint International Workshop: Nuclear Technology and Society – Needs for Next Generation, Berkeley, California, January 6-8, 2008.

[40] Xiong, J., et. al., Numerical Analysis of Droplet Impingement Using the Moving ParticleSemi-Implicit Method, Journal of Nuclear Science and Technology, 47:3 (2012), p.314-321.

[41] Zuo, W. and Chen, Q. 2009. Real time or faster-than-real-time simulation of airflow in buildings, Indoor Air, 19(1), 33-44.

[42] Zuo, W. and Chen, Q., Fast and informative flow simulations in a building by using fast fluid dynamics model on graphics processing unit, Building and Environment 45 (2010) 747–757.

[43] Jin, M., et. al., Simulating buoyancy-driven airflow in buildings by coarse-grid fast fluid dynamics, Building and Environment 85 (2015) 144-152.

[44] Harlow, F. H., et. al., Relativistic Fluid Dynamics Calculations with the Particle-in-Cell Technique, Journal of Computational Physics 20, p.119-129 (1976).

[45] Cook, T. L., et. al., PIC Calculations of Multiphase Flow, J. of Comp. Physics 41, p. 51- 67 (1981)

[46] Xie, J., et. al., Eulerian-Lagrangian method for three-dimensional simulation of fluidizedbed coal gasification, Advanced Powder Technology 24 (2013) p.382–392

[47] Couet, B., Buneman, O., Simulation of Three-Dimensional Incompressible Flows with a Vortex-in-Cell Method, Journal of Computational Physics 39, p.305-328 (1981).

[48] Hejlesen, M. M., et. al., Turbulence modelling in the two-dimensional vortex-in-cell method, Computational Science and Engineering Laboratory, ETH Zurich, Switzerland.

[49] Kudela, H., Regucki, P., The Vortex-in-cell method for the study of three-dimensional vortex structures, Tubes, Sheets and Singularities in Fluid Dynamics, Fluid Mechanics and ItsApplications Volume 71, 2002, p. 49-54.

[50] Guo, Z; Shu, C, Lattice Boltzmann Method and Its Application in Engineering, ISBN:

978-981-4508-29-2, World Scientific Publishing Company, March 2013.

[51] Succi, S, The Lattice Boltzmann Equation for Fluid Dynamics and Beyond, ISBN-13: 978-0198503989, Oxford University Press, Oxford, New York, 2001

[52] Liao, Q, Chien Jen, T, Application of Lattice Boltzmann Method in Fluid Flow and HeatTransfer, Chapter 2, Computational Fluid Dynamics Technologies and Applications, 2011.

[53] Chen, S., Doolen, G. D., Lattice Boltzmann Equation for Fluid Flows, Annu. Rev. Fluid Mech. 1998. 30:329-64.

[54] Yuan, P., Laura, S., A thermal Lattice Boltzmann Two-Phase Flow Model and Its Application to Heat Transfer Problems – Part 1. Theoretical Foundation, Journal of Fluid Engineering, Vol. 128, p.142-150, January 2006.

[55] Rosdzimin1, A. R. M., et. al., Simulation of Mixed Convective Heat Transfer Using Lattice Boltzmann Method, International J. of Automotive and Mechanical Eng. Volume 2, p.130-143, 2010.

[56] Taghilou, M. , Hassan, R. M., Lattice Boltzmann model for thermal behavior of a dropleton the solid surface, International Journal of Thermal Sciences 86 (2014) 1-11.

[57] Zhou, W.N. , et. al. A lattice Boltzmann simulation of enhanced heat transfer of nanofluids,International Communications in Heat and Mass Transfer 55 (2014) p. 113–120.

[58] Gevelera, M., et. al. A Simulation Suite for Lattice-Boltzmann based Real-Time CFD Applications Exploiting Multi-Level Parallelism on modern Multi- and Many-Core Architectures, J. of Computational Science, Vol. 2, Issue 2, May 2011, p. 113–123

[59] Hosain, M. L., Bel-Fdhila, R., Daneryd, A., Multi-jet impingement cooling of a hot flat steel plate, Energy Procedia 61(2014) p.1835 – 1839.