3-DIMENSIONAL ANALYSIS OF DAM BREAK FLOW USING ANSYS

A DISSERTATION

SUBMITTED IN PARTIAL FULFILLMENT OF THE REQUIREMENTS
FOR THE AWARD OF DEGREE
OF

MASTER OF TECHNOLOGY

IN

HYDRAULICS AND WATER RESOURCES ENGINEERING

Submitted By

PUNJ PRATAP SINGH

(Roll No. 2K16/HFE/14)

Under the supervision of

PROF. (DR.) BHARAT JHAMNANI

DEPARTMENT OF CIVIL ENGINEERING

Delhi Technological University

(Formerly Delhi College of Engineering)

Bawana Road, Delhi-110042

JULY,2018

DELHI TECHNOLOGICAL UNIVERSITY

(Formerly Delhi College of Engineering)

Bawana Road, Delhi-110042

CANDIDATE'S DECLARATION

I, Punj Pratap Singh, Roll No. 2K16/HFE/14 of M.Tech. Hydraulics and Water Resources Engineering, hereby declare that the project Dissertation titled **"3-DIMENSIONAL ANALYSIS OF DAM BREAK FLOW USING ANSYS"** which is submitted by me to the Department of Civil Engineering, Delhi Technological University, Delhi in partial fulfillment 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 work has not been used for the award of any Degree, Diploma Associateship, Fellowship or other similar title or recognition.

Place, Delhi, INDIA **PUNJ PRATAP SINGH**

Date:

DELHI TECHNOLOGICAL UNIVERSITY

(Formerly Delhi College of Engineering)

Bawana Road, Delhi-110042

CERTIFICATE

I hereby certify that the Project Dissertation titled **"3-DIMENSIONAL ANALYSIS OF DAM BREAK FLOW USING ANSYS"** by **Punj Pratap Singh**, Roll No. **2K16/HFE/14**, Department of Civil Engineering, Delhi in partial fulfillment of the requirement for the award of the degree of Master of Technology, is a record of the project work carried out by the student under my supervision. To the best of my knowledge, this work has not been submitted in parts or full for any Degree or Diploma to this University or elsewhere.

Place, Delhi, India

Date:

Prof. (Dr.) BHARAT JHAMNANI

Professor, Civil Engineering

Delhi Technological University

Delhi, INDIA

ACKNOWLEDGEMENT

I am deeply grateful to Delhi Technological University for giving me the chance to seek after my Master's degree with all essential facilities. I might want to express my hearty and sincere thanks to my project supervisor Professor Dr. Bharat Jhamnani whom affectionate and excellent supervision has helped me to do my project work. Without his valuable guidance, advice and instructions, this work would not have been successful and timeous. Apart from inestimable academic support gained from him, he is also a professional person, hardworking and humble to everyone. He is a model to me and I owe him much respect.

I would like to thank Dr. Rakesh Kumar Arya for his advice and generosity to discuss different challenges of my research project. Besides I would like to thanks to my father and mother who despite the distance has always been the source of strength towards the accomplishment of this thesis. Their love is too immense to express adequately in words but I owe them a very special gratitude. This is the case also with my friends who gave me enormous encouragement and support in one way or another along this journey.

I extend my deep-felt thanks to my postgraduate colleagues for their companionship during these two years. I acknowledge also each and every one who has contributed to the success of this thesis.

PUNJ PRATAP SINGH

M.Tech (HFE), DTU

Roll NO. 2K16/HFE/14

ABSTRACT

Dams have been recognized for giving power which is the type of renewable energy, for surge insurance, and for making water accessible for agriculture and human needs. Dam break analysis is very crucial for investigating future effects caused to human life and property by the sudden release of large volume of water to the inundation area of a dam. The society gets profited in many ways from the dams but what will happen if dam fails. The consequences are devastating to the society, causes loss of human life due to short warning time available and extensive damage to properties. At the point when a dam is broken, catastrophic flooding will happen as the appropriated water escapes through the breach and streams into the downstream valley which may causes destruction in the downstream area. So, the safety of downstream area is one of the most important aspects during the planning and designing of dam. It is always assumed that large magnitude of flood wave is generated due to failure of dam and inundates large area along the downstream portion of river. Every well-constructed as well as proposed dams need to be examined for the possibility of dam break because even with advanced technology, failure cannot be fully rooted out based on the huge level risks associated with it. With the analysis of velocity profiles, pressure variation and turbulence effect in the downstream locations, we can reduce the vulnerabilities of dam break flood.

The primary motivation behind the study was to examine an unsteady dam break flow. The work displayed here comprise of numerical study on dam break streams. Analyses were focused to assemble extensive information on an unsteady 3D Dam break streams. Dam break simulations were done by using a computational fluid dynamics package, ANSYS FLUENT. The 3D numerical simulations of dam break flow are

done using the large eddy simulation (LES) and the k-epsilon turbulence models and volume of fluid method is used for tracking the free surface of the dam break flow. Dam failure was simulated in ANSYS FLUENT. The gate was not specifically formed, it was just defined as a face without any named boundary conditions. The non-defined face will be taken as a sudden release dam break by the fluent.

CONTENTS

Candidate's Declaration	II
Certificate	III
Acknowledgement	IV
Abstract	V
Contents	VII
List of Tables	IX
List of Figures	X
List of Symbols and Abbreviations	XII
CHAPTER 1 INTRODUCTION	**1**
1.1 OVERVIEW	1
1.1.1 Definition of Dam	1
1.1.2 Dam Break Phenomenon	1
1.2 HISTORY OF DAM FAILURES	2
1.3 NEED FOR DAM BREAK MODELLING	4
1.4 TYPES OF DAM FAILURES	5
1.4.1 Hydraulic Failure	5
1.4.2 Seepage Failure	7
1.4.3 Structural Failure	8
1.5 NUMERICAL MODELLING	9
1.5.1 Ansys	10
1.6 OBJECTIVE OF STUDY	10
1.7 ORGANISATION OF THESIS	11
CHAPTER 2 LITERATURE REVIEW	**12**
2.1 OVERVIEW	12
2.2 PREVIOUS WORK DONE ON DAM BREAK ANALYSIS	12
2.2.1 Theoretical Studies	14
2.2.2 Experimental Studies	14
2.2.3 Numerical Studies	15
2.3 MOTIVATION	18

CHAPTER 3 METHODOLOGY **20**
3.1 OVERVIEW 20
3.2 NUMERICAL MODEL 20
3.3 TURBULENCE MODELLING 21
3.3.1 Turbulence Models 21
3.3.2 Large Eddy Simulation 22
3.3.3 K- Epsilon Model 22
3.4 VOLUME OF FLUID MODEL 23

CHAPTER 4 NUMERICAL SIMULATION **24**
4.1 FRAME WORK OF SIMULATION IN ANSYS FLUENT 24
4.1.1 Preprocessing 25
4.1.2 Setup Physics 28
4.2 MODEL SETUP 30

CHAPTER 5 RESULT AND DISCUSSIONS **32**
5.1 OVERVIEW 32
5.2 SIMULATION OF 3-D DAM-BREAK FLOWS USING LES AND K –E TURBULENCE MODELS 32
5.3 RESULTS 33
5.3.1 Bottom Pressure 34
5.3.2 Velocity 38
5.3.3 Water Surface and Velocity Profiles 45
5.3.4 Eddy Viscosity 50

CHAPTER-6 CONCLUSIONS **52**
6.1 OVERVIEW 52
6.2 CONCLUSIONS 52
6.3 SCOPE FOR THE FUTURE WORK 53

REFERENCES **54**

LIST OF TABLES

Table 4.1: Dimensions of the Dam..25

Table 4.2: Details of Mesh...27

Table 4.5: Measurement locations of dam..30

LIST OF FIGURES

Figure 1.1: Dam failure by overtopping.. 6

Figure 1.2: Erosion of soil of downstream face. 7

Figure 1.3: Failure of dam due to piping through dam body. 7

Figure 1.4: Failure due to piping. 8

Figure 4.1: Geometry of Dam.. 26

Figure 4.2: Meshing of Dam model... 27

Figure 4.3: The boundary condition: Pressure outlet. 29

Figure 4.4: Boundary condition: walls. 29

Figure 4.6: Wired frame of the Dam …... 30

Figure 5.1: Contours of the bottom pressure at time t = 5.0 s....................................... 34

Figure 5.2: Variation of bottom pressure with time at upstream locations..................... 35

Figure 5.3: Variation of bottom pressure with time along the gate................................ 36

Figure 5.4: Variation of bottom pressure with time at the downstream locations…..... 37

Figure 5.5: Velocity vectors at time t = 3.75 s…………………………………………39

Figure 5.6: Variation of depth average velocity at upstream location of

dam.(A& B) 45……………………………………………………….……..40

Figure 5.7: Variation of depth average velocity at gate location of dam. (C& G)....... 41

Figure 5.8: Variation of depth average velocity at downstream location of dam 42

Figure 5.9: Variation of velocity with the length of the dam at time t= 1.25 s ……......43

Figure 5.10: Variation of velocity with the length of the dam at time t= 2.5 s……..….43

Figure 5.11: Variation of velocity with the length of the dam at time t= 3.75 s, 5 s

and 7.5 s …………………………………………………………………..... 44

Figure 5.12: Comparison of w- velocity profiles (stream wise velocity) at location A

using LES and k-epsilon model at t=1.25 s…………………………….…...46

Figure 5.13: Comparison of u- velocity profiles (lateral velocity), at location A

using LES and k-e model at t=1.25 s……………………………………46

Figure 5.14: Comparison of w- velocity profiles (stream wise velocity), at location 11

using LES and k-e model at t=1.25 s.……….…………………………... 47

Figure 5.15: Comparison of u- velocity profiles (lateral velocity), at location 11

using LES and k-e model at t=1.25 s……………………………………47

Figure 5.16: Comparison of w- velocity profiles (stream wise velocity), at location 14

using LES and k-e model at t=1.25 s.………………….……….............. .48

Figure 5.17: Comparison of u- velocity profiles (lateral velocity), at location 14 using

LES and k-e model at t=1.25 s…………………………………………....48

Figure 5.18: Variation of water depths with the length of the dam at different locations

at t =1.25s……………………………………………………………….49

Figure 5.19: Variation of eddy viscosity with time. .... .51

LIST OF SYMBOLS AND ABBREVIATIONS

CFD	Computational Fluid Dynamics
Cn	Courant Number
FVM	Finite Volume Method
LES	Large Eddy Simulation
F(x)	Any Function of x
F'(x)	Derivative of f(x)
H	Depth of Water
Re	Reynolds Number
T	Representation of time
Ux	Derivative of U with respect to x
Ut	Derivative of U with respect to t
U	Velocity of flow along horizontal direction
V	Velocity of flow along vertical direction
VOF	Volume of fluid
X	Horizontal dimension
Y	Vertical dimension

CHAPTER-1

INTRODUCTION

1.1 OVERVIEW

1.1.1 Definition of Dam

A Dam is a barrier set across a waterway or stream to store or back off the stream, which makes an upstream reservoir. These store water can be used for the various purposes. Failure of dam results in quick moving flood waves in the downstream valley with ruinous outcomes including fatalities, property misfortunes, and destruction of infrastructure.

1.1.2 Dam Break Phenomenon

The development of dams in streams can give significant advantages, for example, the supply of drinking and irrigation water and additionally the era of electric power and flood assurance; however the outcomes which would bring about the occasion of their disappointment could be cataclysmic. They differ significantly relying upon the degree of the immersion zone, the measure of the population at danger, and the measure of warning time accessible.

Dam break might be condensed as the partial or disastrous failure of a dam prompting the uncontrolled arrival of water. Such an occasion can majorly affect the area and groups downstream of the breached structure. A dam break may bring about a high flood wave going along a valley at entirely high speeds.

1.2 HISTORY OF DAM FAILURES

Kaddam Project Dam, Andhra Pradesh, India

Worked in Adilabad, the dam was a composite structure, earth fill and/or rock fill and gravity dam. It was 30.78 m high and 3.28 m wide at its peak. The dam was overtopped by 46 cm of water over the peak, in spite of a free board allowance of 2.4 m that was given, bringing on a noteworthy breach of 137.2 m wide that happened on the left bank. The dam failed in August 1958.

Kaila Dam, Gujarat, India

The Kaila Dam in Kachch, Gujarat, India was built amid 1952 - 55 as an earth fill dam with a height of 23.08 m over the river bed and a crest length of 213.36 m. The capacity of full store level was 13.98 million m^3. The foundation was made of shale. In spite of a freeboard recompense of 1.83 m at the ordinary supply level and 3.96 m at the greatest repository level the energy dissipation devices initially failed and later the dike broken down because of the feeble foundation bed in 1959.

Kodaganar Dam, Tamil Nadu, India

This dam in the India, was built in 1977 on a tributary of Cauvery River as an earthen dam with controllers, with five vertical lift shades each 3.05 m wide. The dam was 15.75 m high over the deepest foundation, having a 11.45 m height over the stream bed. A 2.5 m free board over the most extreme water level was given. The dam failed because of overtopping by flood waters which streamed over the downstream slants of the embankment and breach the dam along different spans.

Machhu II (Irrigation Scheme) Dam, Gujarat, India

This dam was worked close Rajkot in Gujarat, India, on River Machhu in August, 1972, as a composite structure. It comprised of a masonry spillway in river area and earthen banks on both sides. The dam failed on August 1, 1979, due to unusual flood and lacking spillway limit. Resulting overtopping of the bank brought on lost

1800 lives. A greatest depth of 6.1 m of water was over the crest and within two hours, the dam failed.

Panshet Dam: (Ambi, Maharashtra, India, 1961 - 1961)

Panshet Dam it supplies drinking water to Pune. Panshet Dam burst in its first year of putting away water on 12 July 1961, when the dam wall burst, on account of the total absence of obligatory reinforced cement concrete (RCC) strengthening, causing enormous flooding in Pune. An expected 1000 individuals died from the subsequent flood.

Khadakwasla Dam (Mutha, Maharashtra, India, 1864 - 1961)

The Khadkawasla Dam, close to Pune in Maharashtra, India was developed in 1879 as a masonry gravity dam, established on hard rock. It had a height of 31.25 m over the stream bed, with an 8.37 m depth of foundation. The failure of the dam happened as a result of the break that created in Panshet Dam, upstream of the Khadkawasla reservoir. The upstream dam discharged a huge volume of water into the downstream store during a period when the when the Khadkawasla reservoir was already full. This brought on overtopping of the dam since inflow was much over the configuration flood.

Tigra Dam: (Sank, Madhya Pradesh, India, 1917 - 1917)

This was a hand set workmanship (in time mortar) gravity dam of 24 m height, built with the end goal of water supply. A depth of 0.85 m of water overtopped the dam over a length of 400 m. Two major squares were substantial pushed away. The failure was because of sliding.

Teton Dam, Teton River canyon, Idaho, USA, NA – 1976

The development started in April, 1972, and the dam was finished on November 26, 1975. The dam was outlined as a zoned earth and rock fill bank, having slants of 3.5 H: 1 V on the upstream and 2 H: 1 V and 3 H: 1 V on the downstream, a height over the bed rock of 126 m, and a 945 m long crest. The dam had a stature of 93 m, a crest width of

10.5 m. The bank material comprised of clayey silt, sand, and rock pieces taken from excavations and burrow areas of the stream's gulch zone. The dam failed on June 5, 1976, the reason for failure was ascribed to piping progressing at a fast rate through the body of the bank. The essential reason for disappointment was viewed as a mix of geographical components and design choices, which taken together permitted the inability to happen. Various open joints in abutment rock and lack of more appropriate materials for the impervious zone were called attention to by the board as the primary driver for the failure of the dam.

Malpasset Dam

An arch dam of height 66 m, with 22 m long crest at its crown. At the point when the breakdown happened, the dam was subjected to a record head of water, which was just around 0.3 m beneath the most elevated water level, coming about because of 5 days of extraordinary precipitation. The failure happened as the arch breached, as the left abutment gave away. The left projection moved 2 m on a level plane with no remarkable vertical movement. The water stamps left by the wave uncovered that the arrival of water was practically on the double. 421 lives were lost and the harm was evaluated at 68 million US dollars.

Baldwin Dam

This earthen dam of height 80 m, was developed for water supply, with its primary earthen embankment at northern end of the pool, and the five minor ones to cover low lying ranges along the border. The failure happened at the northern bank segment. The V-shaped failure was 27.5 m deep and 23 m wide. The harms were assessed at 50 million US dollar.

1.2 NEED FOR DAM BREAK MODELLING

The principal European Law on dam break was presented in France in 1968 after the prior Malpasset Dam failure. In India, Risk evaluation and disaster management plan has been made an obligatory necessity while submitting application for ecological freedom in admiration of stream valley ventures. Planning of Emergency Action Plan after itemized dam break study has turned into a major segment of dam safety programme of India.

The compelling way of dam break floods implies that stream conditions will far surpass the size of most characteristic flood occasions. Under these conditions, stream will carry on distinctively to conditions expected for typical river flow modelling and areas will be immersed, that are not regularly considered. This makes dam break modelling a different study for the risk management and emergency activity arrangement.

1.4 TYPES OF DAM FAILURES

Dam Failure an uncontrolled release of impounded water due to structural deficiencies in dam. Like the greater part of engineering structures, earth dams may fail because of flawed design, improper construction and poor maintenance.

The different reasons for failure might be delegated,

1.4.1 Hydraulic failure
1.4.2 Seepage failure
1.4.3 Structural failure

1.4.1 Hydraulic Failure

Hydraulic records for more than 40% of earth dam failure and might be because of one or a greater amount of the accompanying:

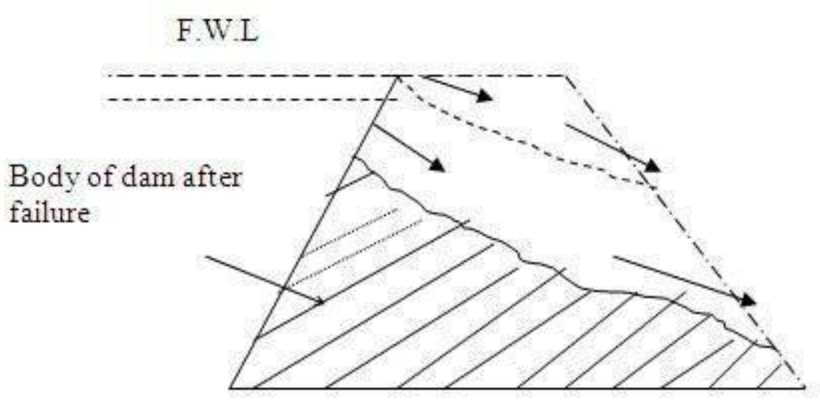


Fig 1.1: Dam failure by overtopping.

By overtopping: When free board of dam or spillway capacity is inadequate, the surge water will pass over the dam and wash it downstream.

Erosion of downstream toe: The toe of the dam at the downstream side might be dissolved because of i) substantial cross-currents and flow from spillway containers, or ii) tail water. At the point when the toe of downstream is disintegrated, it will prompt failure of dam. This can be counteracted by giving a downstream slant pitching or a riprap up to a height over the tail water depth Also, the side wall of the spillway ought to have adequate height and length to anticipate plausibility of cross flow towards the earth embankment.

Erosion of upstream surface: During winds, the waves created close to the top water surface may cut into the dirt of upstream dam face which may bring about slip of the upstream surface prompting failure. For forestalling against such disappointment, the upstream face ought to be ensured with stone pitching or riprap.

Erosion of downstream face: During heavy rains, the streaming precipitation water over the downstream face can erode the surface, making gullies, which could prompt failure. To avert such failures, the dam surface ought to be legitimately maintained; all cuts filled on time and surface all around grassed. Berms could be given at reasonable heights and surface very much depleted.

Fig 1.2:Erosion of soil of downstream face.

1.4.2 Seepage Failure

Seepage always happens in the dams. If the magnitude is within design limits, it may not hurt the dependability of the dam. In any case, if drainage is concentrated or uncontrolled past points of confinement, it will prompt to failure of the dam. Taking after are a portion of the different sorts of seepage failures.

Piping through dam body: At the point when drainage begins through poor soils in the body of the dam, little channels are framed which transport material downstream. As more materials aretransported downstream, the channels shine greater and greater which could prompt wash out of dam.

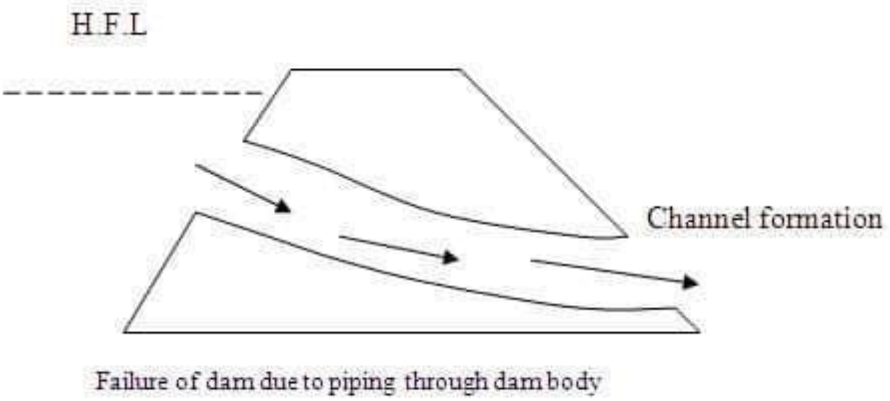


Fig 1.3: Failure of dam due to piping through dam body.

Piping through foundation: When highly permeable depressions or crevices or strata of gravel or coarse sand are available in the dam foundation, it might prompt over leakage. The accumulated leakage at high rate will disintegrate soil which will bring about expansion stream of water and soil. Therefore, the dam will settle or sink prompting failure.

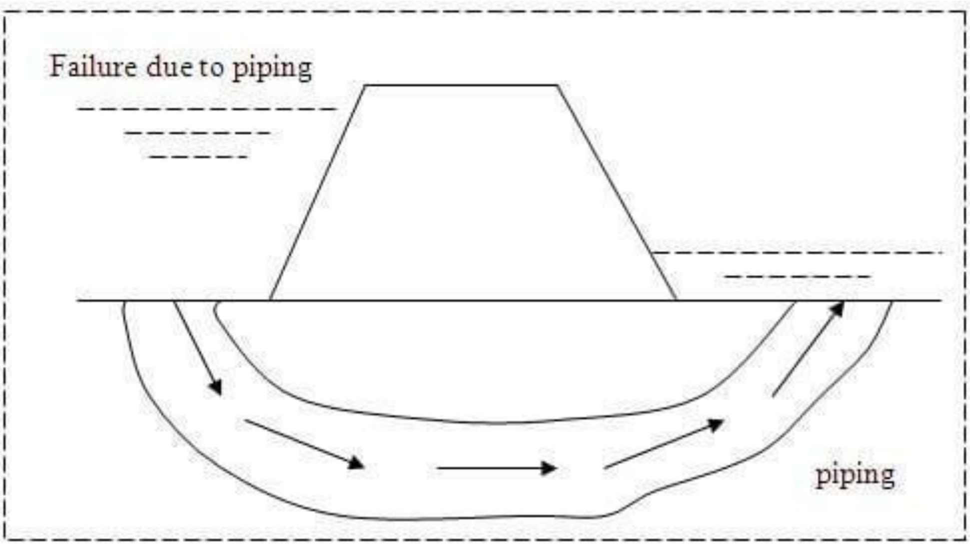


Fig 1.4: Failure due to piping.

Sloughing of downstream side of dam: The procedure of disappointment because of sloughing begins when the downstream toe of the dam gets to be soaked and begins getting eroded, bringing about little slump or slide of the dam. The little slide leaves a relative steep face, which likewise gets to be immersed because of drainage furthermore droops again and frames more unstable surface. The procedure of immersion and slumping keeps, prompting failure of dam.

1.4.3 Structural failure

Around 25% of failure is ascribed to structural failure, which is primarily because of shear failure creating slide along the inclines. The failure might be because of:

Slide in embankment: When the slopes of the banks are excessively steep, the embankment may slide bringing about disappointment. This may happen when there is a sudden drawdown, which is basic for the upstream side as a result of the advancement of to a great degree high pore pressures, which diminishes the shearing quality of the soil. The downstream side can likewise slide particularly when dam is full. Upstream embankment failure is not as genuine as downstream failure.

Slide in foundation: When the Foundation of an earth fill dam is made out of fine silt, mud, or comparable delicate soil, the entire dam may slide because of water push. In the event that creases of fissured rocks, for example, soft clay, or shale exist underneath the foundation, the side push of the water pressure may shear the entire dam and cause its failure. In such disappointment the highest point of the dam gets cracked and slides down, the lower slant faces moves outward and forms huge mud waves close to the dam heel.

Earthquake may bring about the accompanying sorts of inability to earth fill dams;

1. Splits may create in the center wall, bringing about spillages and piping disappointment.

2. Moderate waves may set up because of shaking of reservoir base, and dam may fail because of over topping.

3. Settlement of dam which may decrease free board creating disappointment by over topping.

4. Sliding of characteristic slopes making harm dam and its appurtenant structures.

5. Shear slide of dam.

6. The sand underneath foundation may condense.

7. Failure of incline pitching.

1.5 NUMERICAL MODELLING

Computational Fluid Dynamics (CFD) is a PC based numerical examination tool. The fundamental standard in the use of CFD is to examine fluid stream in-subtle element by illuminating an arrangement of nonlinear governing conditions over the region of enthusiasm, in the wake of applying determined limit conditions. A stage has been taken to do numerical investigation on a Dam break. The utilization of computational liquid elements was another vital part for the fulfilment of this anticipates since it was the fundamental tool of simulation. In general, CFD is a tool which simulates very accurately in various applications like fluid flow, heat transfer, mass transfer and chemical reactions.

1.5.1 Ansys

There are a variety of CFD programs available that possess capabilities for modelling multiphase flow. Some common programs include ANSYS and COMSOL, which are both multiphysics modelling software packages and FLUENT, which is a fluid flowspecific software package. A CFD is a popular tool for solving transport problems because of its ability to give results for problems where no correlations or experimental data exist and also to produce results not possible in a laboratory situation and also useful for design since it can be directly translated to a physical setup and is cost-effective.

1.6 OBJECTIVE OF STUDY

From the literature review it is derived that there is less work involved in simulation in ANSYS Fluent that represents a dam break flow. So this research includes the simulation of dam break flow in software using different turbulence model.

The goals of the study are to pick up bits of knowledge into unsteady stream fields of dam-break stream, to gain top notch information utilizing new estimation systems for approval of numerical models, and to study dam-break streams utilizing a non-depth averaged methodology.

The particular destinations are:-

1. To study dam break flow using ANSYS fluent, the Volume of Fluid (VOF) approach is utilizing for free surface tracking.

2. To investigate the effect of turbulence in dam break flow using the large eddy simulation (LES) and k-epsilon method.

3. Conduct the simulation of Dam break in the ANSYS FLUENT, which is a computer based tool and gather the information with respect to velocity profiles, bottom pressure variation, water surface rise and surface velocity.

4. Finally compare both large eddy simulation method and k-epsilon method results.

1.7 ORGANISATION OF THESIS

This thesis consists of 6 chapters which include chapter-1 Introduction, chapter-2 Literature review, chapter-3 methodology, chapter-4 Numerical simulation, chapter-5 Result and Discussion, chapter-6 Conclusions and scope of work, Chapter-7 – References.

1. Chapter-1 consists of general introduction of the present study and different software used.

2. Chapter-2 consists of past research work done the study area.

3. Chapter-3 consists of basic methods which are used to simulate the dam break flow.

4. Chapter-4 consists of the details regarding Numerical simulations and the procedure to be followed in simulating the ANSYS fluent software.

5. Chapter-5 consists of results obtained from ANSYS fluent simulations.

6. Chapter-6 consists of conclusions of the present work and the scope of the future work to be carried out on the present study area.

7. Chapter-7 consists of references used for the present study.

CHAPTER-2

LITERATURE REVIEW

2.1 OVERVIEW

Researcher and experts are doing and moving closer an extensive variety of courses for comprehension numerical technique in water resources planning. The frameworks are all that much complex because of examination of a dam break stream. However, among them some are clear and predict the surge wave by multiplication process. Specialists and experts are moving nearer a predominant made system regulated by altering the previous one. In computational fluid dynamics, the examination used to address the flood of water after dam break is known as Shallow water question that is resolved kind of N-S numerical proclamation. A rate of the works and examination of dam break stream using numerical methods done by particular scientists are discussed underneath.

2.2 PREVIOUS WORK DONE ON DAM BREAK ANALYSIS

David R. Basco (1989) researched on lone waves and lab scale dam break flood waves proliferating in one dimensional channelare looked at as depicted by finite difference equation to the de Saint Venant and Boussinesq equation. He watched that the de Saint Venant and boussinesq conditions gave fundamentally the same as results aside from brief period waves beneath around 100sec. he presumed that de Saint Venant condition are more satisfactory to catch the material science of these occasions.

J.V. Soulis, et. al. (1991) portrayed around a second-cream sort of total variation diminishing (TVD) restricted difference wore down two dimensional improvement of water on a dry bed as a result of prompt dam break. They assessed it numerically and in like manner tenacious state stream arrangements are broke down to acknowledge the accuracy of proposed numerical arrangement.

P. Brufau and P. Garcia-Navarro(1993)taken a shot at numerical showing of shallow water stream in two estimations that is poor down through dam break tests. Free surface stream in channels can be depicted experimentally by the shallow water structures of numerical explanations. These questions have been discretized using a strategy in light of unstructured Delaunay triangles and associated with the entertainment of two-dimensional dam break streams.

Mohapatra P.K, et al. (1999) worked on numerical calculations for the investigation of dam break stream utilizing two-dimensional stream conditions as a part of a vertical plane. The time assessment of flow depth at the dam site and the evolution of the pressure distribution are explored for both wet and dry bed conditions.

Kratutich (2004) succeeded to concentrate numerically on leakage of earthen dams. He inferred that drainage and thermal distribution are of the same principles. So he did pressure driven investigation with thermal technique at ANSYS programming.

Mimi Das Saikia and Arup Kumar Sarma (2006)were produced a numerical model for recreating dam break flood and connected for dissecting flood circumstance because of the prompt speculative failure of the proposed dam in the river Dibang. They did two distinctive methodologies, in one approach, the expectations are made by embracing a computational channel, which considers the entire flood plain downstream of the dam and alternate considers just improved illustrative stream channel.

Francesca et al. (2008) performed experimental and two dimensional numerical examination for four tests concerning quickly fluctuating stream affected by the sudden evacuation of a floodgate door. In 95% of the neighbourhood examinations with exploratory information gained through a trials a most extreme deviation of 20% was observed.

Manciolaet al. (2010) performed numerical analysis of free surface streams provoked by a dam break differentiating the shallow water approach with totally three-dimensional propagations. The complete arrangement of Reynolds-Averaged Navier-

Stokes (RANS) mathematical statements coupled to the volume of fluid (VOF) framework.

Kamanbedast and A. Delvari (2012) endeavour soil stability of dam has been finished with utilizing ANSYS. They contrasted their outcomes and Geo Studio programming results. They inferred that the ascertained estimations of leakage rate is almost equivalent in both ANSYS and Geo Studio strategy. They got noteworthy contrast in two programming is identified with well-being element and they presumed that ANSYS answer is more adequate. The dam is at reasonable circumstance as per the software results, simply vertical settlement at centre zone ought to be concentrated progressively and perfectly.

Saqib Ehsan and Walter Marx (2014) researched on the Mangla dam is one of the biggest earth fill and rock dams in the world, situated on Jhelum river in Pakistan. The Erosion based overtopping failure of Mangla dam with raised conditions has been examined by utilizing MIKE 11 dam break module.

2.2.1 Theoretical Studies

Ritter (1892) inferred an analytical answer for the quick dam-break stream up a flat and friction less channel expecting an unbounded length for both store and channel. Dressler (1952) and Whitham (1955) incorporated the impact of bed resistance in dam-break stream examinationand got expressions for the velocity and height of the wave-front (Mohapatra, 1998). The Ritter arrangement was reached out by Stoker (1957) to the instance of wet-bed condition downstream of the dam. Hunt (1983, 1984) determined an analytical solution of dam-break stream by considering finite length repositories. Chanson (2006) extended the Ritter answer for the instance of dam-break stream over a frictionless inclining bed.

2.2.2 Experimental Studies

The scholastic enthusiasm for the demonstrating of dam-break streams comes from the test of precisely anticipating the shock condition created by dam-break stream. The Army Corps of Engineers led dam-break tests up 1960, in which a staff bar was put at the edge of a flume and a video camera recorded the water level. This analysis was a spearheading work in quantitative dam-break thinks about, where the spread, shape and velocity of the wave were broadly concentrated on (Schmidgall and Strange, 1960a). Trial displaying of dam-break stream has for the most part included estimations of the free surface variety in 1-D and 2-D stream (e.g. Schmidgall and Strange, 1960b; Miller and Chaudhry, 1989; Aziz, 2000; Soares-Frazao and Zech, 2002, 2007). Various exploratory works additionally included roundabout estimation of the velocity field by different picture examination methods (e.g. Soares-Frazao and Zech, 2002; Eaket et al., 2005; Aureli et al., 2008; Aleixo et al., 2011). Direct estimation of stream velocity in a dam-break investigation is uncommon. Fraccarollo and Toro (1995) led direct estimation at various locations and additionally depth and pressure estimations in a halfway dam breach model. The work of Fraccarollo and Toro (1995) presents novel information set on 3-D dambreak stream. Fraccarollo and Toro (1995) performed point velocity estimations at various areas utilizing a current meter, subsequently giving time series data of point velocity. Stansby et al. (1998) and Janosi et al. (2004) led explores different avenues regarding dry and wet bed conditions downstream of the dam. Stansby et al. (1998) watched a level stream and mushroomlike elements separately, in their tests with dry quaint little inn bed downstream conditions.Janosi et al. (2011) acquired velocity profiles in 2-D dam-break streams utilizing a particle tracking velocimetry (PTV) strategy. Be that as it may, subsequent to the strategy rely on upon discovery of seeding in pictures, the close bed velocity profiles in their outcomes were not generally all around determined.

2.2.3 Numerical Studies

C.Zoppu and S. Roberts(1999) depicted around a numerical model for the arrangement of the two-dimensional dam–break issue. This model is in light of a second-request surmised Riemann solver with a van Leer sort limiter is utilized to unravel the shallow water wave mathematical statement on a Cartesian network. The shallow water comparisons incorporate source terms which represent imperviousness to the stream and the impact of the bed slant or slope.

J.S. Wang, H.G. Ni and Y.S. He(2000) depicted around a second-crossover kind of aggregate variety reducing (TVD) limited contrast plan is examined for understanding dam-break issues. The plan is based upon the first-arrange upwind plan and the second-arrange Lax-Wendroff plan, together with the one-parameter limiter or two-parameter limiter. A similar investigation of the plan with diverse limiters connected to the Saint Venant equations for 1D dam break waves in wet overnight boardinghouse bed cases demonstrate a few distinctions in numerical execution. An ideal chose limiter is gotten. The present plan is stretched out to the 2D shallow water comparisons by utilizing an administrator part procedure.

S.Farzin, M. Alizadeh and Y. Hassanzadeh(2002) considered about numerical reenactment of precarious one-dimensional dam break stream utilizing TVD MacCormack Scheme. Dam break sensation is still of vital essential issue in the field of water powered designing. Anticipating the basic conditions because of dam-break streams shows more field studies prerequisite. The MacCormack numerical plan is a traditional second request unequivocal plan for the recreation of flimsy dam-break streams. It is no doubt understood that traditional second request plans show oscillatory conduct close discontinuities and can produce or keep up a stun in the arrangement.

Dongfang Liang, Binliang Lin and Roger A. Falconer(2006) presented a proficient numerical plan for comprehending the SWEs(shallow water mathematical statements) in ecological stream; which incorporates the expansion of five-point symmetric aggregate variety diminishing(TVD) term to the corrector venture of the standard MacCormak plan. The discretisation of the moderate and non-traditionalist types of the SWEs prompts the same limited distinction plan when the source term is discretised in a certain manner. The bed grinding is incorporated by neighborhood water profundity either expressly or certainly with reproduction of wetting and drying process all the while.

Changhong Hu and Makoto Sueyoshi(2009) presented two novel numerical reckoning systems coming about because of numerical reenactment of dam break test for firmly nonlinear wave body cooperation issues, for example, boat movements in unpleasant oceans and coming about green water affect on deck taking into account Cartesian matrix strategy.

C. Biscarini,S. Di Francesco and P. Manciola(2010) have taken a shot at numerical recreations of free surface streams prompted by a dam break contrasting the shallow water approach with completely three- dimensional reproductions. The arrangements are taking into account the arrangement of the complete arrangement of Reynolds-Averaged Navier-Stokes (RANS) mathematical statements coupled to the volume of fluid(VOF) system.

Szu-Hsien Peng(2012) worked on 1D and 2D numerical displaying for tackling dam break stream issues utilizing limited volume system which purposed to model the stream development in a romanticized dam break design. One dimensional and two-dimensional movement of a shallow stream more than an unbending slanted bed is considered. The reproductions are accepted by the correlation with flume tests. Unstable dam-break stream development is discovered to be sensibly very much caught by the model. This idea can be further created to the numerical estimation of non-Newtonian liquid or multilayer liquid stream.

Hamid Reza Vosoughifar, Azam Dolatshah and Seyed Kazem Sadat Shokouhi (2013) recreated both wet and dry bed dam break issues. A high determination limited volume system (FVM) was utilized to unravel the one dimensional (1D) and two-dimensional (2D) shallow water mathematical statements (SWEs) utilizing an unstructured Voroni network lattice. Numerical Flux at cell interfaces are figured by the strong nearby Lax- Friedrichs (LLxF) METHOD.

Tsang-Jung Chang, Hong-Ming Kao, Kao-Hua Chang, Ming-Hsi Hsu(2014) taken a shot at numerical reproduction of shallow water dam tear streams in open channels utilizing smoothed molecule hydrodynamics . a work less numerical model is proposed to research shallow water dam tear streams in 1d open channels. The numerical model is to comprehend the shallow water equations (SWE) in light of smoothed particle hydrodynamics (SPH). The numerical affectability investigation is initially performed to study the proper SWP number and variable smoothing length through dam break streams in an admired 1D channel with dry/wet beds.

Turbulence modelling of dam-break streams should be possible by one of the accompanying:

(i) Large Eddy Simulation (LES):

In LES, the Navier-Stokes conditions are sifted and extensive scale eddies are determined straightforwardly, while little eddies are modelled.

(ii) Reynolds-Averaged Navier-Stokes (RANS) approach.

The Reynolds averaged Navier- Stokes (RANS) equations describe the transport of the averaged flow quantities, and model the whole scope of the sizes of turbulence bringing about huge lessening in computational expense. The RANS approach e.g., the k epsilon displaying has restrictions including turbulence terminations.

2.3 MOTIVATION

In spite of impressive examination led on dam-break streams, imperative crevices exist in our knowledge of the stream forms. Since dam-break tests include streams that are exceptionally transient and quickly fluctuated, estimations of velocity are not basic. Numerical simulations are frequently performed by tackling the shallow water conditions.

Terrible occasions, for example, dam-breaks, frequently cause broad surge harm to urban and local locations. There has been a significant enthusiasm for numerical demonstrating of these occasions as of late. Be that as it may, far reaching information on overflowed urban ranges are not accessible and shallow water models are frequently connected to think about urban flooding by adding porosity terms to represent the nearness of structures.

Late advances in computational procedures take into consideration determining the 3-D stream field of transient open channel stream by understanding the Navier-Stokes conditions utilizing different turbulence demonstrating choices and following the free surface by vigorous techniques, for example, the Volume of Fluid (VOF) and Level Set strategies.

The present commitment gives new and exhaustive information set on (i) velocity profiles in the upstream reservoir and downstream overflowed territory from 3-D glorified dam-break experiment; (ii) hydrostatic and total pressure, 3-D surface velocity, and water profundity from 3-D dam-break experiments and Dam break simulation in ANSYS software.

CHAPTER 3

METHODOLOGY

3.1 OVERVIEW

In this part, a brief portrayal of the test setup, instruments, estimation methods, and the numerical model are depicted. The 3-D dam-break cases comprised of 3-D tests led in the hydraulics laboratory at the National Institute of Technology Rourkela and simulation of the analysis was also done there only. All simulations were directed utilizing ANSYS FLUENT, a commercial CFD programming.

3.2 NUMERICAL MODEL

In this study, Fluent, a Computational Fluid Dynamics (CFD) tool is utilized for model confirmation, which depends on the three-dimensional type of the Navier Stokes conditions. Computational Fluid Dynamics (CFD), is the branch of fluid mechanics that utilizations numerical strategies and calculations to break down and tackle issues that include fluid streams. The PCs are utilized to perform estimations which required to simulating the collaboration of liquids and gasses with surfaces characterized by boundary conditions. Continuous exploration yields programming which enhances the precision and rate of complex simulation scenarios, for example, transient or turbulent flows. The CFD construct simulation depends in light of the consolidated numerical exactness, modelling accuracy and computational expense.

In general Computational Fluid Dynamics utilizes a finite volume method (FVM). Fluent can use both organized and unstructured systems. In free surface demonstrating, e.g. Volume of Fluid (VOF) (Ferziger and Peric 2002) and height of liquid (HOL), the primary conditions are discretized in both space and time which for the most part requires in transient simulation. Here LES model is utilized for turbulence demonstrating. The LES conditions are discretized in both space and time.In this study the algorithms adopted to solve the combination between pressure and velocity field is PISO, which is used to simulate the transient problems which converges the difficulties in faster way.

GOVERNING EQUATIONS

The separated or Reynolds-Averaged conservation conditions for mass and force for an incompressible liquid can be communicated, individually as (Ferziger and Peric, 2003):

$$\frac{\partial \overline{u_i}}{\partial x_i} = 0$$

And

$$\frac{\partial \rho \overline{u_i}}{\partial t} + \frac{\partial \rho \overline{u_i u_j}}{\partial x_j} = -\frac{\partial \bar{p}}{\partial x_i} + \frac{\partial}{\partial x_i}\left[\mu\left(\frac{\partial \overline{u_i}}{\partial x_j} + \frac{\partial \overline{u_j}}{\partial x_i}\right)\right] + \frac{\partial \tau_{ij}}{\partial x_j}$$

Where u_i and u_j are sifted or Reynolds-arrived at the midpoint of velocities, and are Cartesian direction axes; ρ is the fluid density. The separated or Reynolds-found the middle value of pressure t is the time and μ is the molecular viscosity. The term τ_{ij} signifies the Reynolds stress.

3.3 TURBULENCE MODELLING

Turbulence is an asymmetrical movement which with everything taken into account appears in fluid, liquids, or vaporous, when they stream past strong surfaces or even when neighbouring streams of the same liquid stream past or more than each other. GI Taylor and von Karman, 1937.

Turbulent smooth motion is an unpredictable state of stream in which the diverse amounts show an arbitrary variety with time and space facilitates, so that factually particular typical qualities can be watched. Hinze, 1959.

3.3.1 Turbulence Models

- Large eddy simulation method (LES).
- Detached eddy simulation method (DES).
- Scale – Adaptive simulation method (SAS).
- Reynolds stress (7 equations).
- Transition SST method.
- K- Omega turbulence model.
- K- Epsilon turbulence model.

3.3.2 Large Eddy Simulation

In the Large Eddy Simulation approach, bigger vortexes are determined and littler or sub-lattice scale eddies are modelled permitting preferable constancy over generally approaches. In the present study, the sub grid scale model proposed by Smagorinsky (1963) is utilized.

$$\tau_{ij} - \frac{1}{3}\tau_{kk}^{s}\delta_{ij} = \mu_t\left(\frac{\partial \overline{u_i}}{\partial x_j} + \frac{\partial \overline{u_j}}{\partial x_i}\right) = 2\mu_t\overline{S_{ij}}$$

Where μ_t is the sub-grid viscosity and S_{ij} is the strain rate of the bigger scale or determined field.

The eddy viscosity is modelled as

$$\mu_t = C_s^2\rho\Delta^2 \,|\bar{S}|$$

Where C_S is a model parameter, which is equal to 0.1, Δ is the filter length scale,

And

$$|\bar{S}| = (\overline{S_{ij}S_{ij}})^{1/2}.$$

3.3.3 K- Epsilon Model

K-epsilon (k-∈) turbulence model is the most widely recognized model utilized as a part of Computational Fluid Dynamics (CFD) to simulate mean stream qualities for turbulent stream conditions. It is a two condition model which gives a general depiction of turbulence by method for two transport conditions. The initially transported variable decides the energy in the turbulence and is called turbulent kinetic energy (k). The second transported variable is the turbulent dissipation (∊) which decides the rate of dispersal of the turbulent kinetic energy.

For turbulent kinetic energy k

$$\frac{\partial(\rho k)}{\partial t}+\frac{\partial(\rho k u_i)}{\partial x_i}=\frac{\partial}{\partial x_j}\left[\frac{\mu_t}{\sigma_k}\frac{\partial k}{\partial x_j}\right]+2\mu_t E_{ij}E_{ij}-\rho\epsilon$$

For dissipation ∊

$$\frac{\partial(\rho\epsilon)}{\partial t}+\frac{\partial(\rho\epsilon u_i)}{\partial x_i}=\frac{\partial}{\partial x_j}\left[\frac{\mu_t}{\sigma_\epsilon}\frac{\partial\epsilon}{\partial x_j}\right]+C_{1\epsilon}\frac{\epsilon}{k}2\mu_t E_{ij}E_{ij}-C_{2\epsilon}\rho\frac{\epsilon^2}{k}$$

The eddy- viscosity model in the Reynolds-averaged approach is expressed as

$$\tau_{ij}=\left(\frac{\partial\overline{u_i}}{\partial x_j}+\frac{\partial\overline{u_j}}{\partial x_i}\right)-\frac{2}{3}\rho k\delta_{ij}$$

3.4 VOLUME OF FLUID MODEL

The Volume of Fluid (VOF) is a surface-following method connected to a settled Eulerian network intended for two or more immiscible liquids where the position of the interface between the liquids is of interest. In the VOF model, a solitary arrangement of energy conditions are shared by the liquids and the volume fraction of each of the liquids in each computational cell are followed all through the area. Uses of the VOF model incorporate stratified streams, free surface streams, filling, the movement of large bubbles in a fluid flow, the forecast of plane break, and the tracking of a fluid gas interface (Hirt and Nichols, 1981). Give a part function C a chance to be characterized as the indispensable of liquids trademark capacity in the control volume. On the off chance that the cell is vacant, the estimation of C is 0; if the cell is full, C is 1; and if the interface cuts the cell, then C is somewhere around 0 and 1. The cell esteem, C computes from (Hirt and Nichols, 1981).

CHAPTER 4

NUMERICAL SIMULATION

4.1 FRAME WORK OF SIMULATION IN ANSYS FLUENT

The Numerical simulation process in ANSYS fluent contents different steps and those steps were followed below.

(i) Problem Identification

1. Defining the goals of model.
2. Identify the domain to the model.

(ii) Pre-Processing

1. Creating a Geometry setup.
2. Create and design the grid using mesh operator.

(iii) Solver

1. Solution setup
   - Define the flow condition, for example turbulence flow, laminar flow and viscous flow.
   - Select the materials that are going to be used, specify the phases also and give the Boundary conditions and Operating conditions.

2. Using the specific numerical scheme from different schemes present in solver to discretize the governing equations.

3. Controlling the convergence by iterating the equation till accuracy is achieved.

4. Calculate the solution by solver settings.
   - Solution method

- Solution controls.
- Solution initialization.
- Run calculation.

(iv) Post Processing

1. Visualizing and examine the computed values.
2. Plotting the graphs.
3. Contour drawing.

4.1.1 Preprocessing

In this underlying stride all the vital information which describes the issue is allotted by the client. This involves geometry, the properties of the computational mesh, distinctive models to be utilized, and the amount of Eulerian stages, the time step and the numerical arrangements.

4.1.1.1 Creation of Geometry

The underlying stage in CFD examination is the illumination and creation of computational geometry of the liquid stream district. A predictable edge of reference for direction pivot was reference for production of geometry. Here in direction framework, Z axis related to the direction of fluid flow of dam break, X axis related to the lateral direction of the dam, Y axis is related to the direction parallel to the dam height. The upstream reservoir length is 1m and width is 2 m, height of dam is 0.6 m. the dimensions of downstream reservoir length, width and height are 8 m, 2 m, 0.1 m respectively. The dam site is horizontal and downstream dry condition. The dimensions of the Dam is shown in Table 4.1 and the setup of Dam break model is shown in Figure 4.1.

Table 4.1: Dimensions of the Dam

	Upstream Reservoir	Downstream Reservoir	Gate
Length	2	8	
Width	2	2	0.5
Height	1	0.3	0.95

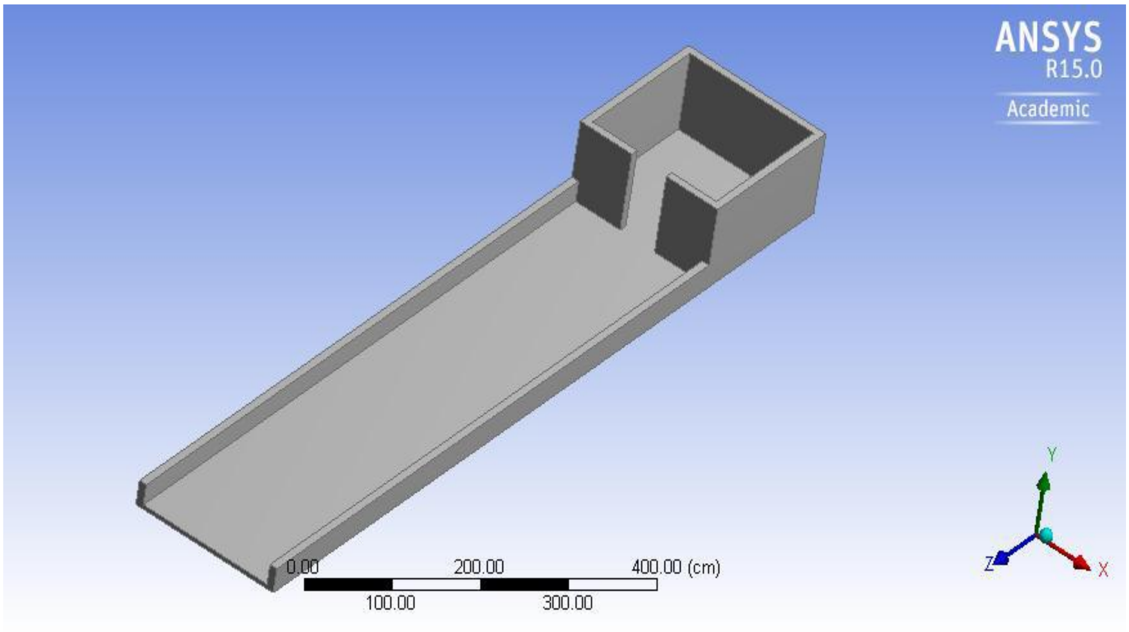


Figure: 4.1. Geometry of Dam

4.1.1.2 Mesh Generation

Second and most critical step in numerical investigation is setting up the grid related with thedevelopment of geometry. The Navier-Stokes Equations are non-linear PDE, which consider theentire liquid domain as a continuum. With a specific end goal to streamline the issue theconditions are rearranged as basic streams have been straight forwardly settled at low Reynolds numbers. The simplification can be made utilizing what is called discretization. The creation of mesh includes discretizing or subdividing the geometry into the cells or small elements at whichthe variables will be processed numerically. By utilizing the Cartesian co-ordinate framework, the liquid stream governing equations i.e. momentum condition, continuity condition aresettled in light of the discretization of domain.

The CFD examination needs a spatial discretization plan and time marching plan. Meshingdivides the whole geometry into finite number of nodes and elements. Generally the domainsare discretized by three diverse ways i.e. Finite element, Finite Volume and Finite Difference Method. In finite element method the domain was divided into number of elements. In finite element method the numerical arrangements are gotten by incorporating the shape work and weighted element in a proper space. This technique is appropriate for both organized and unstructured grid. Be that as it may, in the Finite Volume technique the domain was divided into finite number of volumes. The discretization of the solution is done at center of the volume in the method of finite volume. The details of meshing is shown in below table 4.2, the setup of meshing is shown in Figure 4.2.

Table: 4.2. Details of Mesh

Domain	Nodes	Elements
Fluid	967	3100
Solid	1548	2127
All Domains	2515	5227

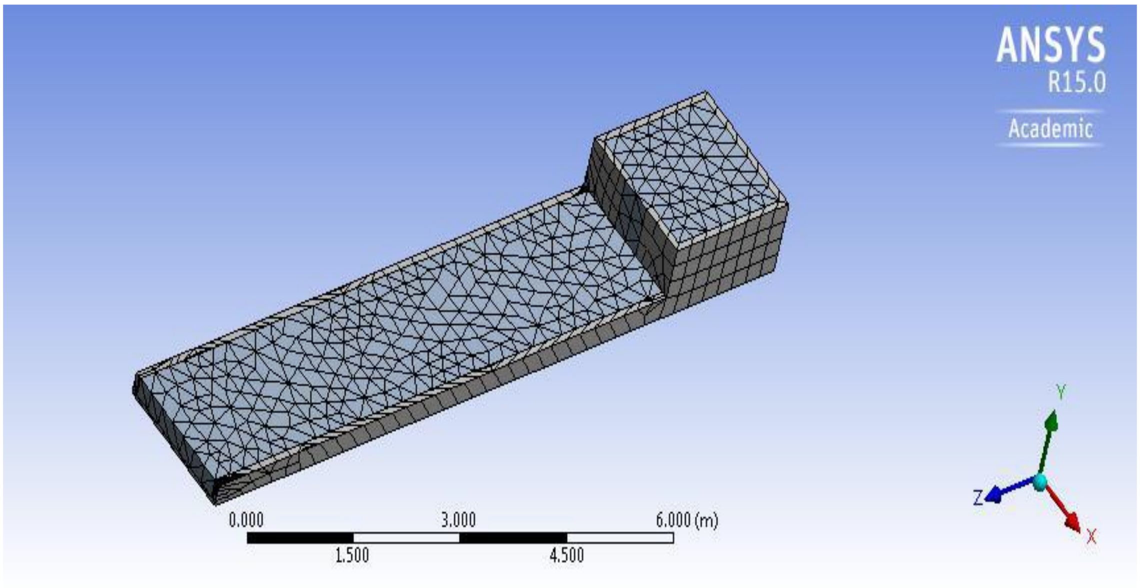


Figure: 4.2. Meshing of Dam model

For transient issues a fitting time step should be indicated. To catch the required features of liquid stream with in a space, the time step ought to be adequately little however not all that much little which may bring about misuse of computational power and time. Spatial and time discretization's are connected, as apparent in the Courant number.

Courant Number

A basis as often as possible used to decide time step size is known as Courant number (Cn). The Courant number prevents the time venture from being sufficiently substantial for data to travel totally through one cell amid one emphasis. For explicit time stepping plans Courant number ought not to be more than 1. For implicit time stepping plans this number might be higher than 1.

4.1.2 SETUP PHYSICS

For a given computational area, boundary conditions are mandatory which can once in a while over determine or under-indicate the issue. As a rule, subsequent to forcing boundary conditions in non-physical area may prompt disappointment of the answer for convergence. It is along these lines critical, to comprehend the significance of very much posed boundary conditions.

4.1.2.1 PRESSURE OUTLET BOUNDARY CONDITION

The dam break simulation was done in ANSYS fluent. After the completion of geometry and meshing of dam model, the boundary condition has to be given. For the creation of instant dam break simulation, top surface and downstream boundary named as pressure outlets. The gate was not given any named boundary condition. The above mentioned as the top surface of upstream reservoir and downstream reservoir was named as pressure outlet that's way the water stored in upstream reservoir creates dam break simulation after some time step and flows water to downstream through the gate portion. The all side portions and boundary of outlet named as walls. The boundary conditions were shown in Figure 4.3 and Figure 4.4.

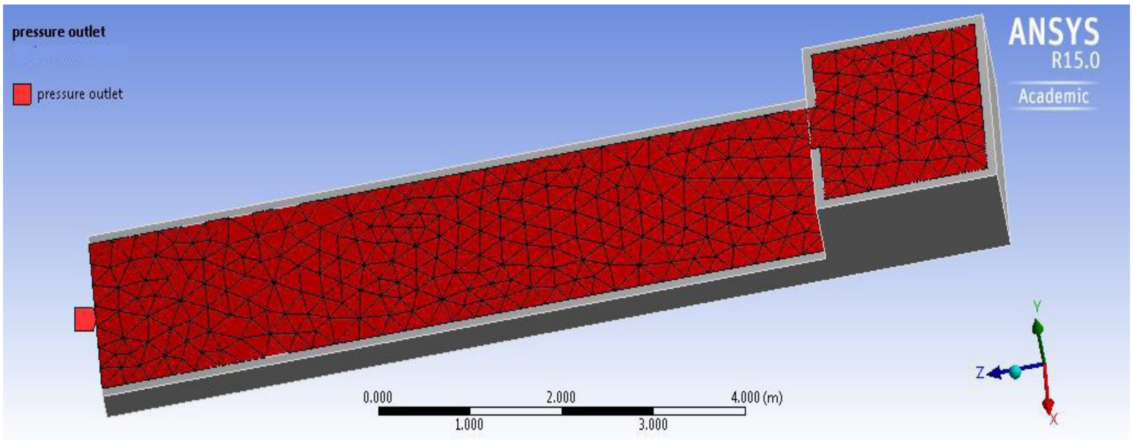


Figure: 4.3. The boundary condition: Pressure outlet

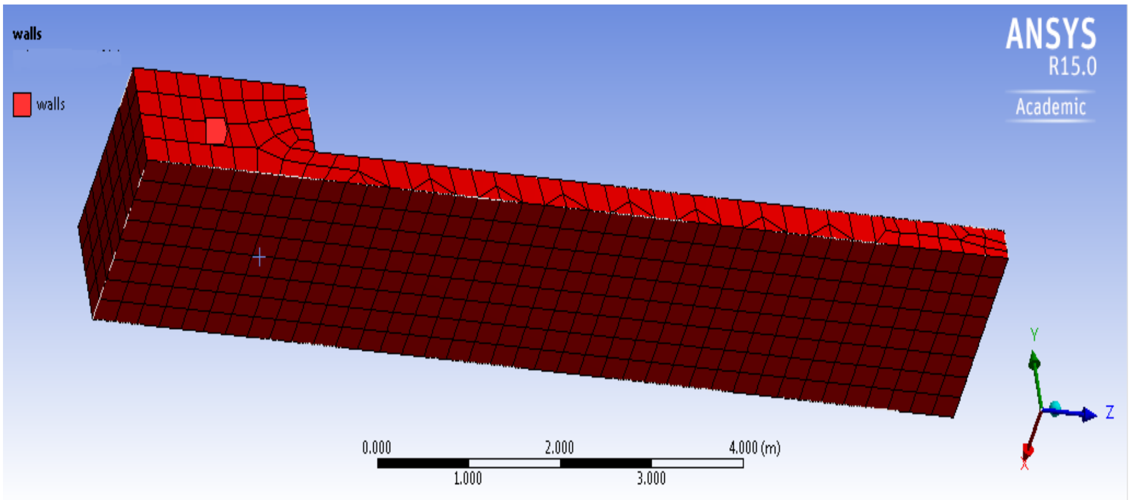


Figure: 4.4. Boundary condition: walls.

4.1.2.2 FREE-SURFACE

For top free surface for the most part symmetry boundary condition is utilized. This determines the shear stress at the divider is zero and the stream wise and lateral velocities of the liquid close to the divider are not impeded by divider erosion impacts as with a no-slip boundary condition. This condition takes after that, no flow of scalar flux happens over the boundary. In this manner, there is neither convective flux nor diffusive flux over the top surface. In executing this condition ordinary velocities are set to zero and estimations of all different properties outside the area are likened to their qualities at the closest node simply inside the space.

4.2 MODEL SETUP

The model setup consisting of different steps, such as geometry of dam model, meshing, schematic diagram of the dam and measurement locations. The geometry and meshing were explained above in detailed. The schematic diagram of dam was shown in Figure 4.5 and the measured locations were shown in Table 4.5.

Table: 4.3 Measurement locations of dam.

Position	A	B	C	D	E	F	G	11	14
X (m)	1	1	1	1	1	1	1.25	0.5	1.5
Y (m)	0.05	0.05	0.05	0.05	0.05	0.05	0.05	0.05	0.05
Z (m)	0.8	1.2	2	3.5	5	7.5	2	2.5	2.8

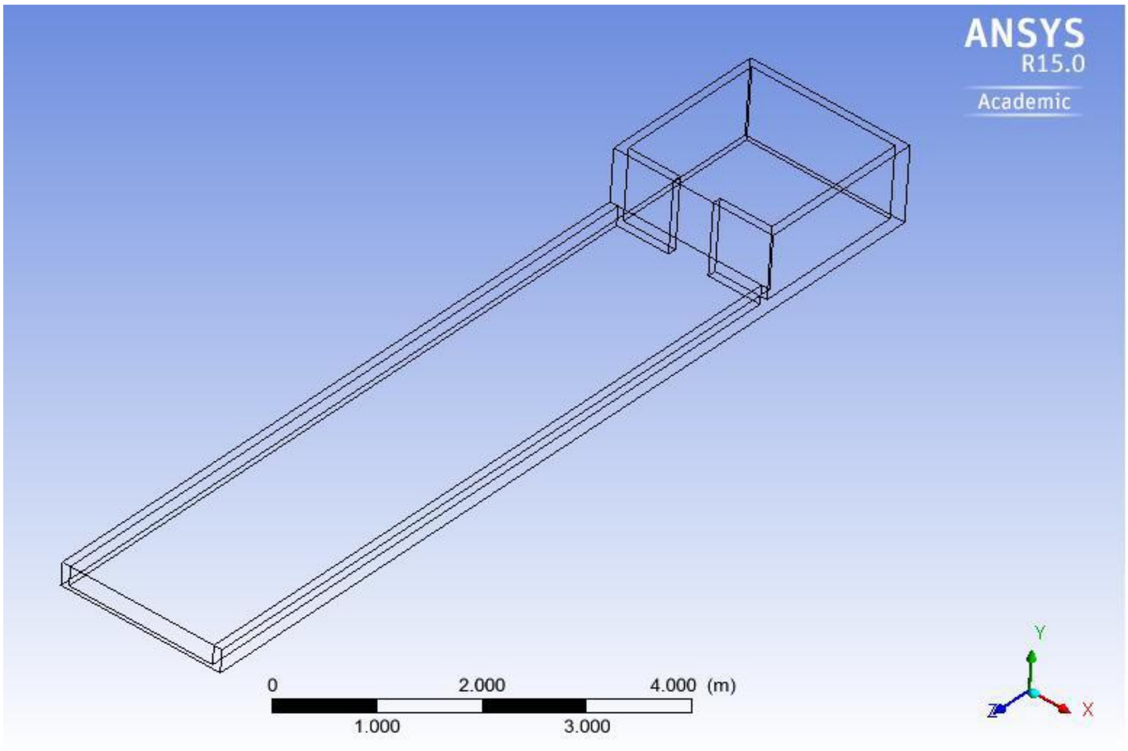


Figure: 4.6. Wired frame of the Dam

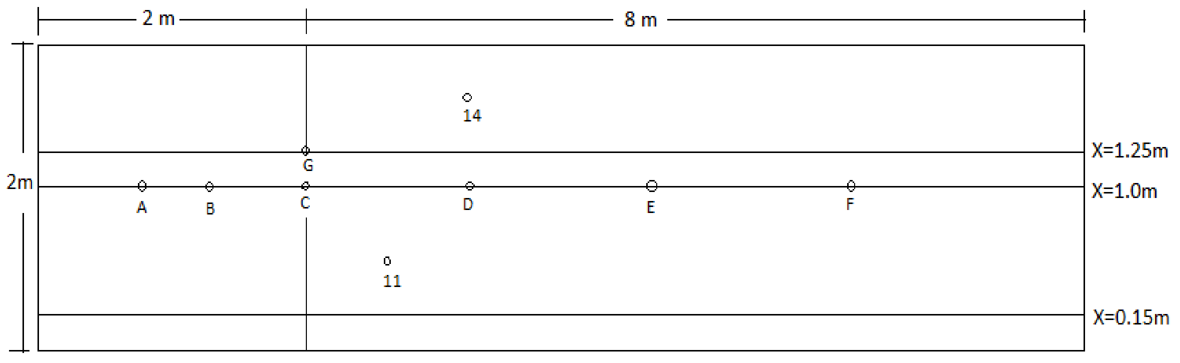


Figure 4.7: Measurement locations.

CHAPTER 5

RESULT AND DISCUSSIONS

5.1 OVERVIEW

From the past research done on the dam break analysis, it was watched that information on 3-D dam-break stream are uncommon. A striking special case is the work of Fraccarollo and Toro (1995), who led a point by point test on partial-breach dam-break flow. The Computational Fluid Dynamics solver FLUENT was used to lead a 3-D simulation of the analysis of Fraccarollo and Toro (1995). Second, trials were led in a moderately extensive setup to get thorough information set on velocity profiles, 3-D surface velocities, water depths, static, and total pressure. Both LES and k - e models were considered for turbulence demonstrating. The VOF model was utilized for surface following as a part of simulating the analysis of Fraccarollo and Toro (1995). It was watched that the k-e model performs to some degree inadequately in predicting the measured data.

5.2 SIMULATION OF 3-D DAM-BREAK FLOWS USING LES AND *k* –*e* TURBULENCE MODELS

In numerical simulation, the LES model and K-epsilon model correlations were made of water depth, bottom pressure, and velocity profiles. The present study showed that, as opposed to the ordinarily held perspective, turbulent impacts can assume a critical part for close fields in a dam break stream. In SWE models, the friction impacts are globalized as bed shear stress parameterized by rubbing laws, for example, Manning or Chezy condition.

MODEL SETUP

The Figure 4.5 shows the schematic and measurement locations of the dam break simulation. The width of upstream reservoir was 2 m, length also 2 m, height of the upstream water was 1 m with gate opening of 0.5 m width. The downstream of the dam was 8 m long, 2 m wide and 1 m deep. The downstream area was primarily taken as dry condition. For little scale reproductions with spotlight on the subtle elements, for example, turbulence blasting and appearance of barrette vortices, and unsteadiness highlights, very fine meshes are utilized. However, in LES displaying of huge scale streams, for example, climatic boundary layer where the point by point expectation of the turbulence elements is not of essential interest, coarse meshes are regularly utilized (e.g. Stoll and Porte-Agel, 2006). In that capacity, there is no all-around acknowledged standard for the determination of grid size in LES modelling. The time step *t*was taken as 0.025 s, that value was based on the courant condition.

In the numerical model, the sides, back, sides encompassing the outlet and the base are characterized as solid walls. The highest point of the flume and downstream outlet are indicated as a pressure outlet. The gate was not specifically created, it was just defined as a face without any named boundary conditions. Fluent will understand the non-defined face as a sudden release dam break.

5.3 RESULTS

Numerical results are used to display the variety of velocity, water surface, and bottom pressure with respect to time. Once the water is discharged from the gate, it proliferates both in downstream and horizontal bearings before leaving at the downstream end. Amid the principal second of stream improvement got utilizing the LES model. The water going through the gate proliferates both in downstream and horizontal bearings. The stream meets on the reservoir side and diverges on the downstream side. The stream extends downstream of the door, comes to and reflects off of the side dividers, and afterward moves towards the downstream end. The stream territory downstream of the dam increments in size with time.

5.3.1 BOTTOM PRESSURE

In the dam break flow simulation pressure was calculated at various locations, such as A, B, C, D, E, F, G (Figure 4.5) using both the LES model K-epsilon model and compared them with respect to time.

The Figure 5.2 shows the bottom pressure variation with time calculated from the large eddy simulation model and K- epsilon model. The bottom pressure at the locations in the upstream reservoir, such as position A and Position B initially had a pressure of 1 m that to slowly decreases with increase in time. At time t=0 there is no pressure near to the gate position C and position G, after the starting of simulation the bottom pressure increase suddenly and reaches the peak value instantaneously.

And further increase of time the bottom pressure was reduced because of water spreads in the downstream and reaches the end point of downstream area. The variation of bottom pressure at upstream and gate position was shown in figure 5.2. and figure 5.3 respectively. The contours of bottom pressure shown in the Figure 5.1.

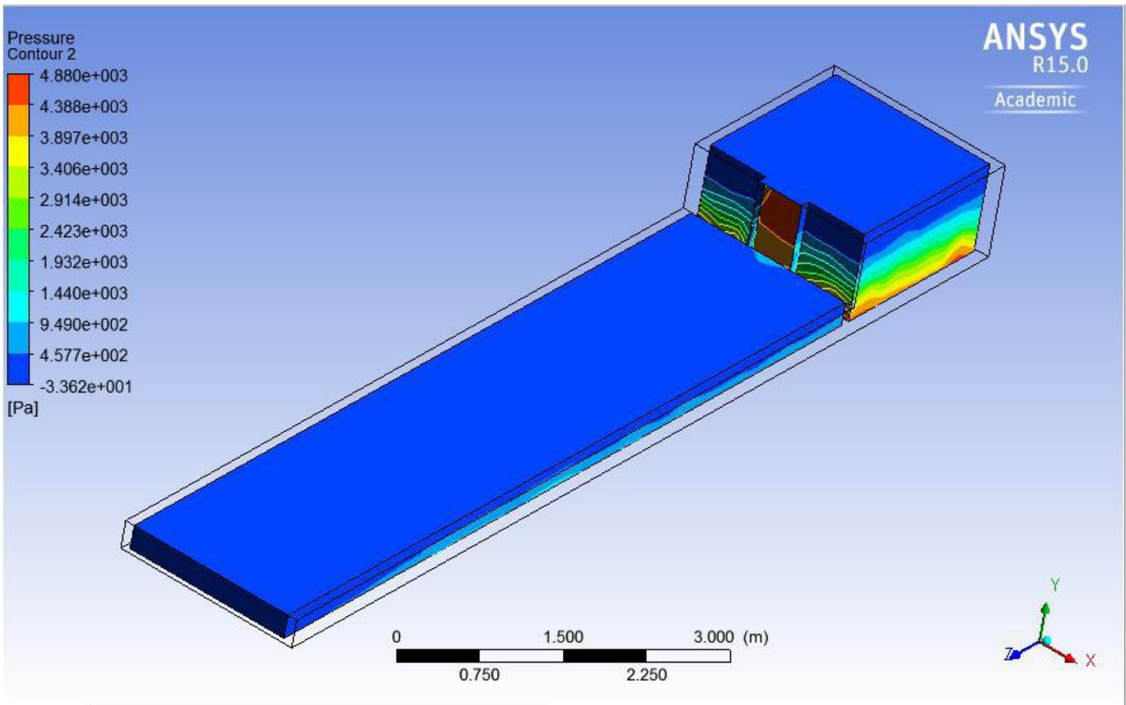
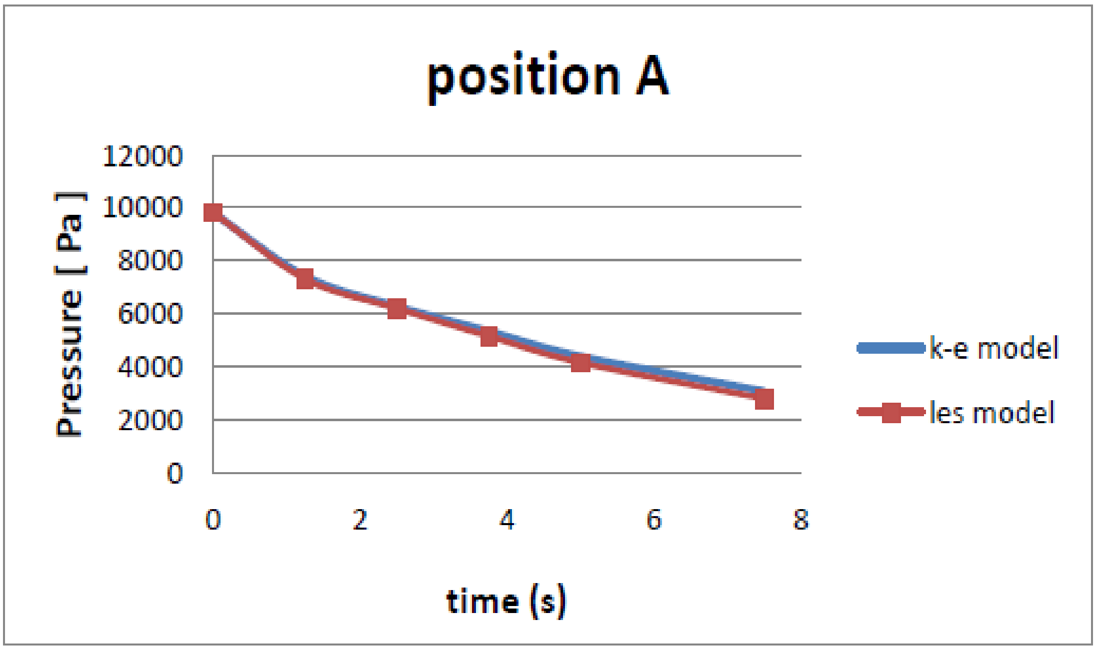
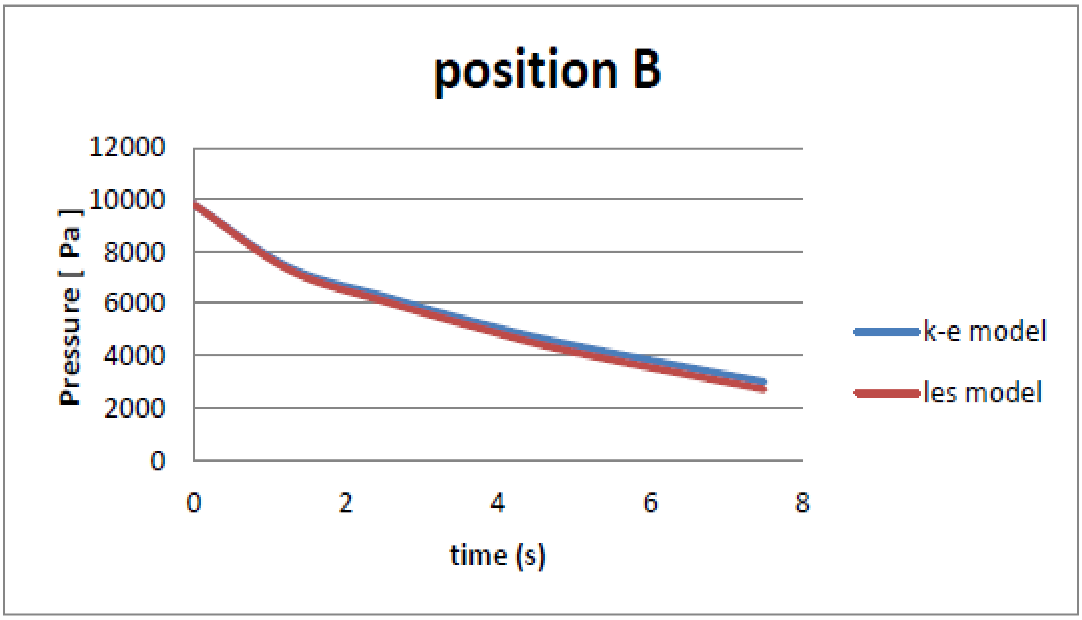


Figure 5.1: Contours of the bottom pressure at time t = 5.0 s

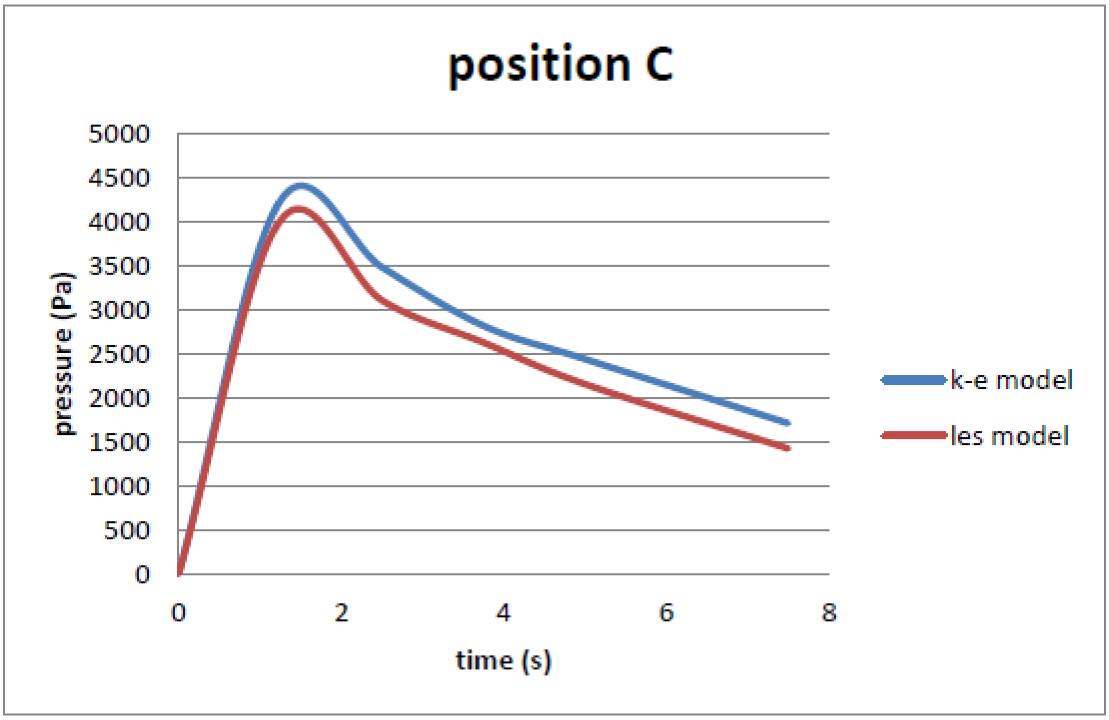


(a)

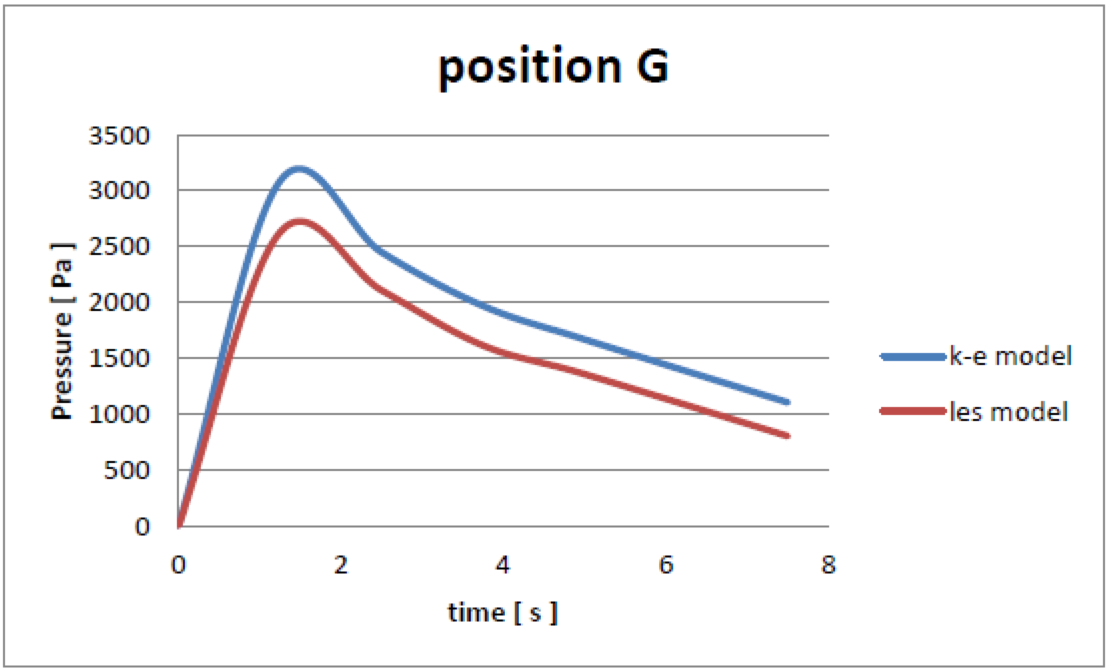


(b)

Figure 5.2: Variation of bottom pressure with time at upstream locations



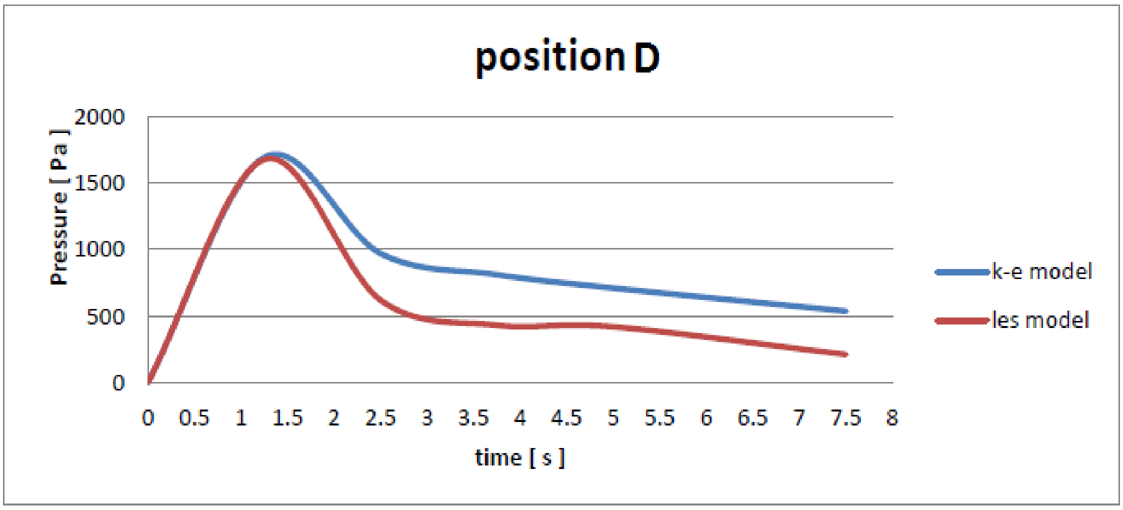
(a)



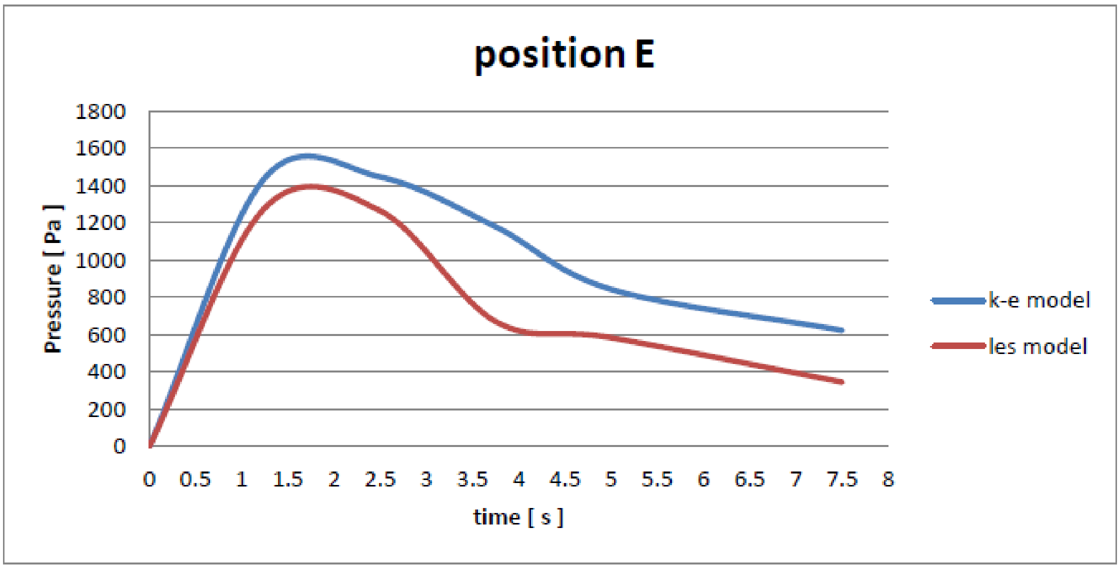
(b)

Figure 5.3: Variation of bottom pressure with time along the gate.

The positions along the downstream are D and E, the bottom pressure at those positions achieving their peak values after some time. We observed the time lag between gate opening and increase in pressure. The peak pressures at near the gate opening positions are significantly higher than the downstream locations. The variations of bottom pressure at the downstream points were shown in Figure 5.4.



(a)



(b)

Figure 5.4: Variation of bottom pressure with time at the downstream locations.

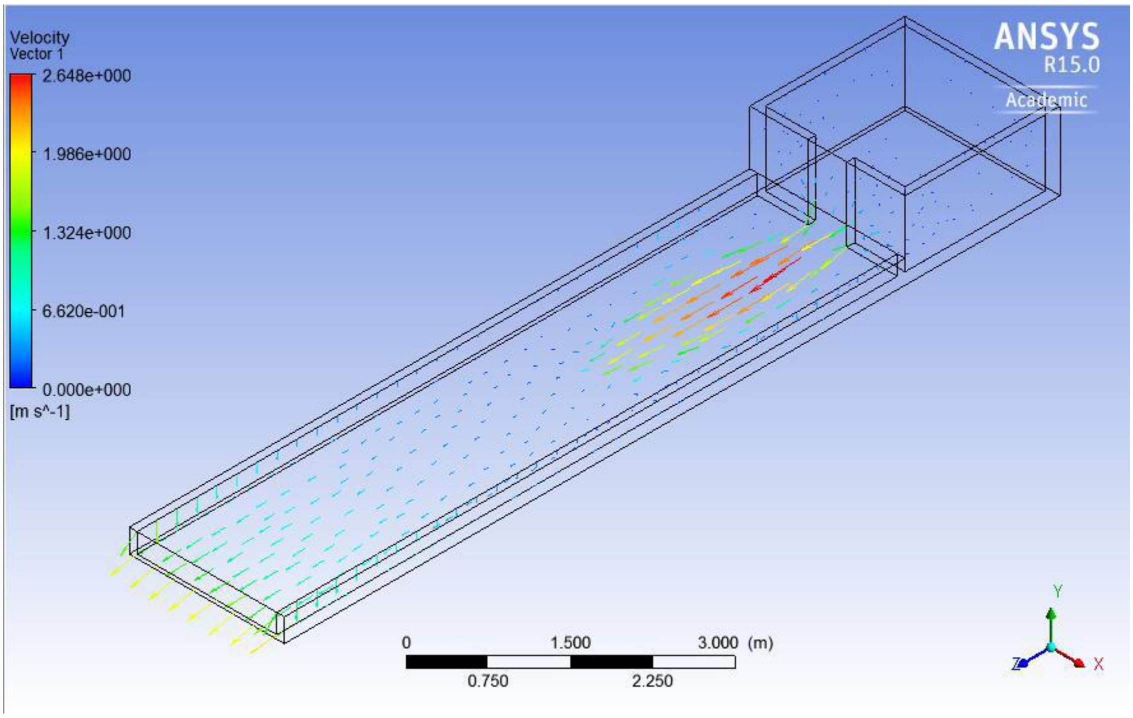
At the upstream locations the large eddy simulation turbulence model and K- epsilon model had the nearly same results. Both the graphs were nearly coincided that was shown in Figure 5.2. Along the gate positions the k- epsilon model had the high values compared to LES model that difference was shown in Figure 5.3.

5.3.2 VELOCITY

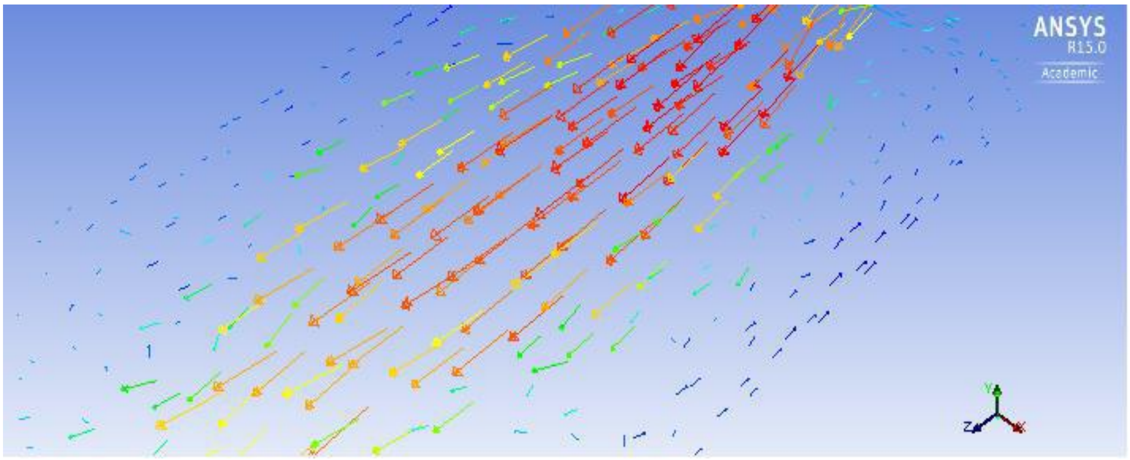
The velocities were measured at various positions like A, B, C, D, E, F, G using ansys fluent. The velocity contours after time of 3.5sec is shown in figure 5.6 The calculation of velocities were done at upstream locations, such as A and B using both the LES and K-e model and is shown in figure 5.6.

The velocities at upstream side increases with respect to time and reached the peak at nearly 2.5 s after the dam break simulation, after that the velocity decreases further increase of time and maintained nearly constant value. At the gate location the velocity reaches peak value within short time and decreases slowly, that was shown in Figure 5.7.

In case of downstream velocities there was a time lag between the opening of gate and increase in velocity. At the position E velocity was nearly zero upto time t= 2 s, then after increases slowly. After the time t=5 s velocity was remained same because the water level was same in both the upstream and downstream of the dam. The variation of downstream velocity with time was shown in Figure 5.8.

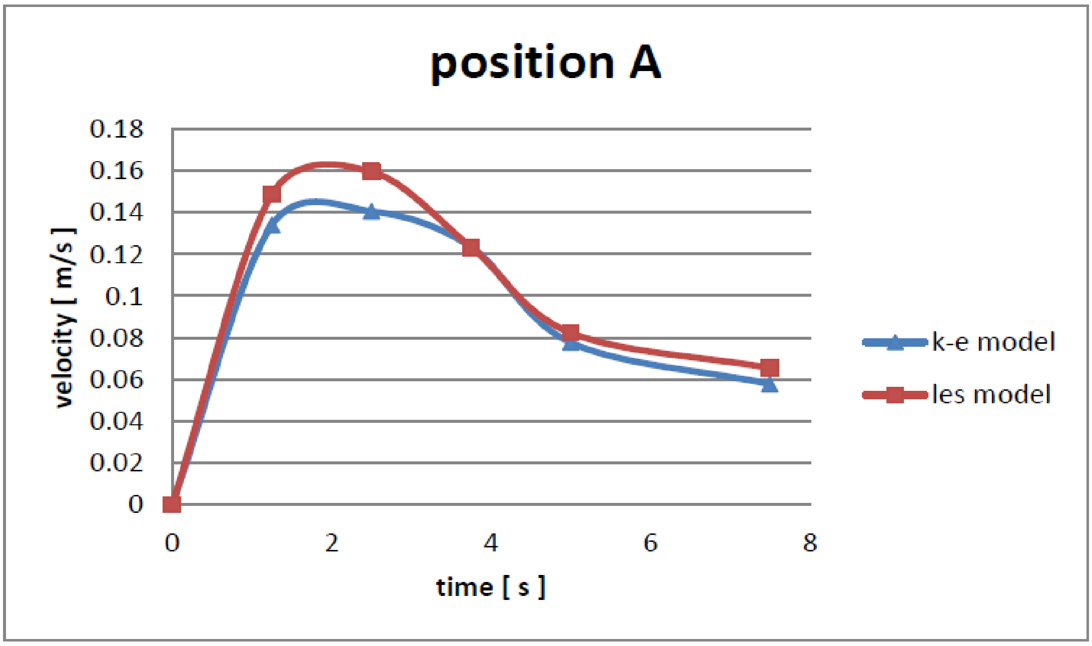


(a)

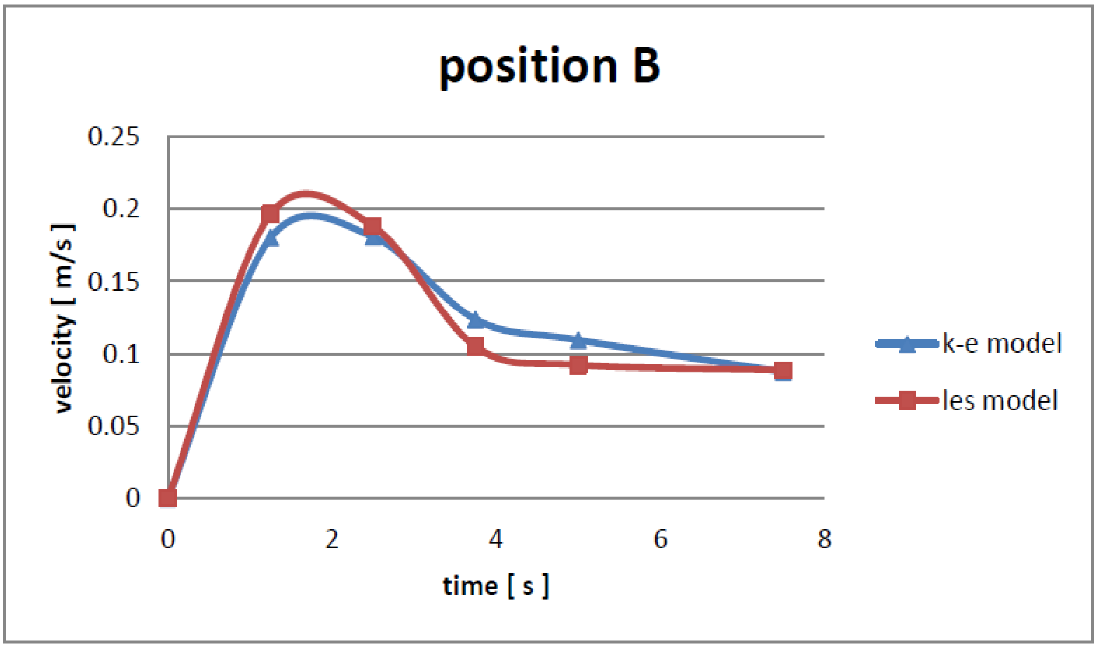


(b)

Figure 5.5: (a) & (b) Velocity vectors at time t = 3.75 s.

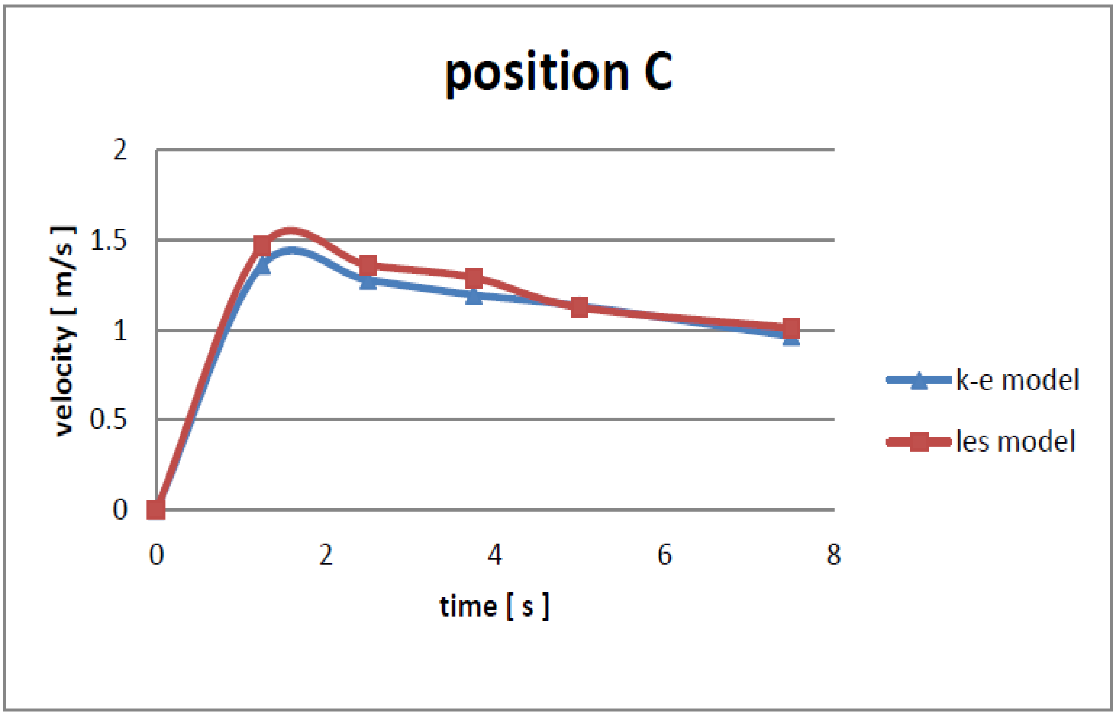


(a)

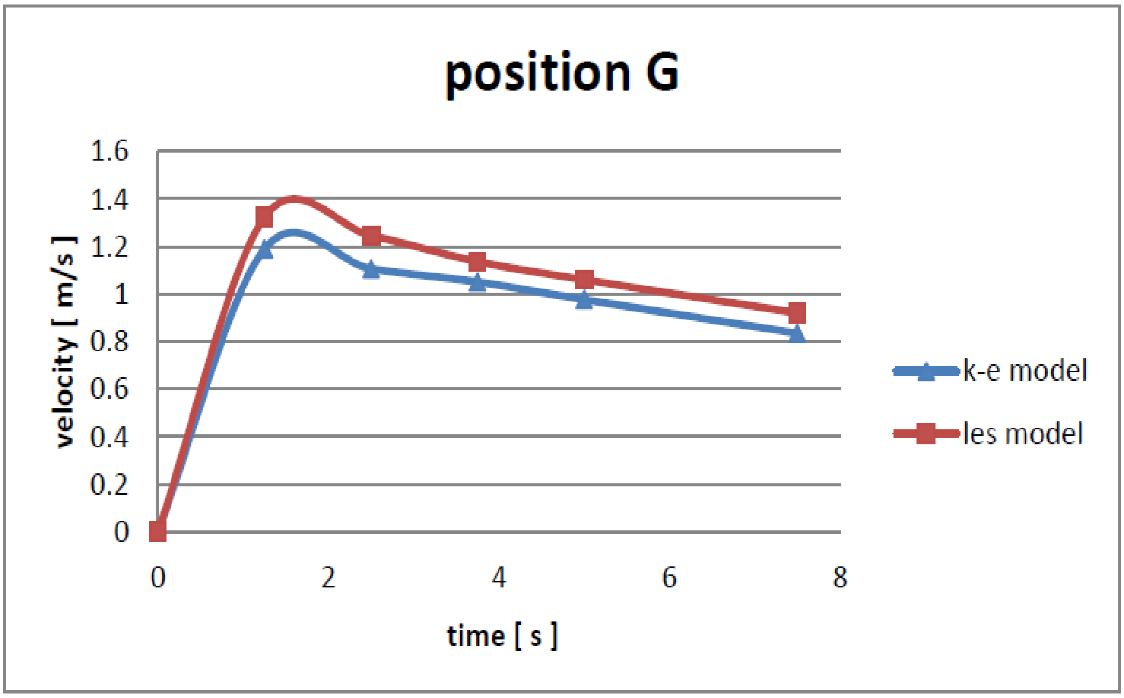


(b)

Figure 5.6: Variation of average velocity at upstream location of dam (A& B)

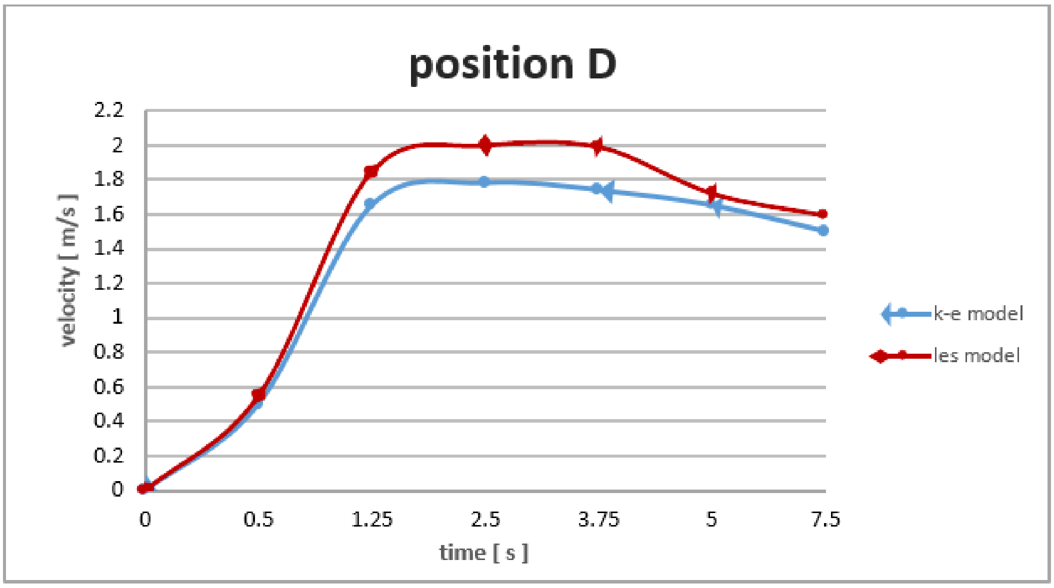


(a)



(b)

Figure 5.7: Variation of average velocity at gate location of dam. (C& G)



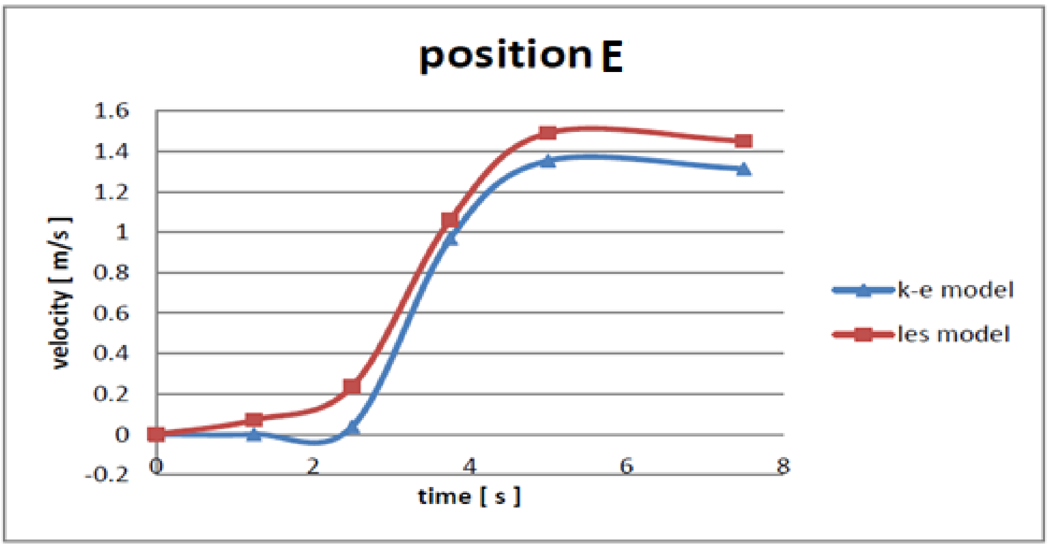


Figure 5.8: Variation of average velocity at downstream location of dam (D & E).

Velocity of the wave increases along the length immediately after the break and decrease with respect to distance after the surge. The variation can be seen in the graph and also through velocity contours. It is around the dam gate the velocity is high due to the breach and properties of the channel which allow the flow to spread into a wide cross section allowing the velocity to reduce along the stream.

Values observed at runtime 1.25 and 2.5 are approximately close, but with any increase of runtime velocities are decreasing. This is mainly due to the emptying of storage volume after the breach and increase in cross section on the downstream.

The below graphs shows the variation at the time t = 1.25s, 2.5 s, 3.75 s, 5.0 s and 7.5 s and compare the large eddy simulation model and k-epsilon model. The variation of velocity along the length of the dam was shown in the following figures.

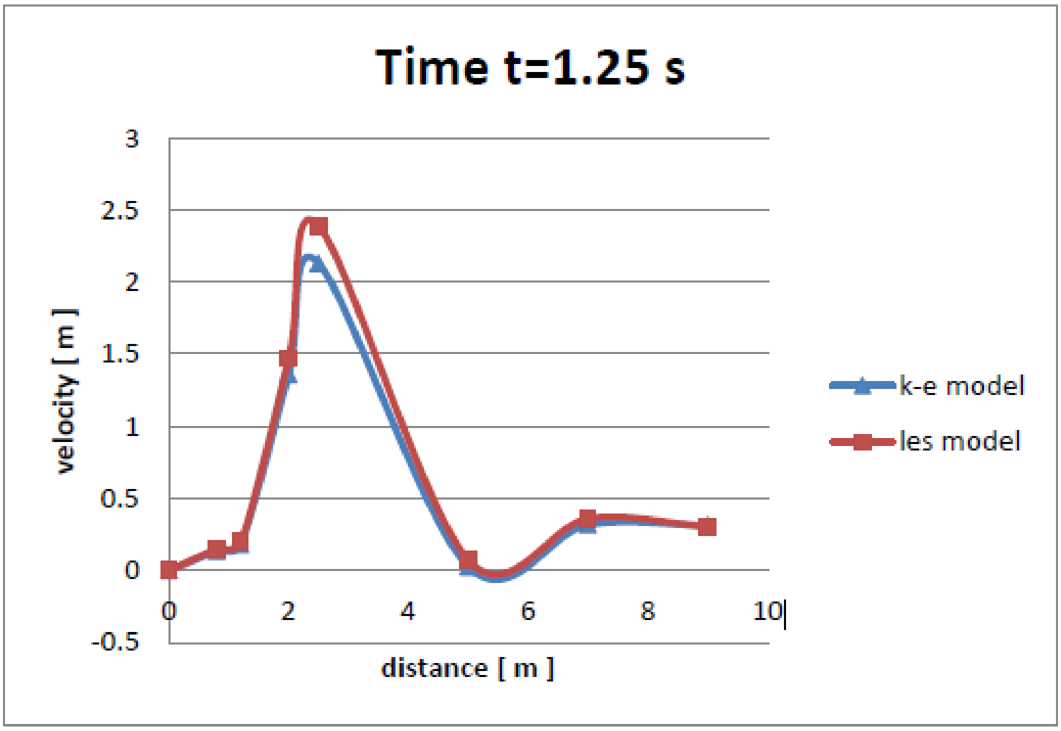


Figure 5.9: Variation of velocity with the length of the dam at time t= 1.25 s.

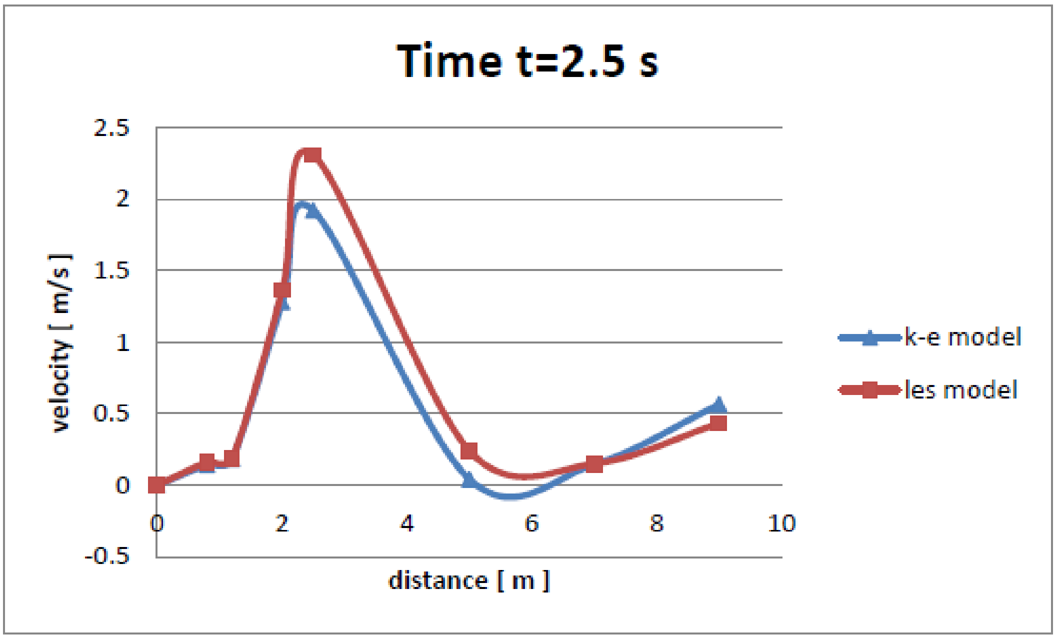


Figure 5.10: Variation of velocity with the length of the dam at time t= 2.5 s.

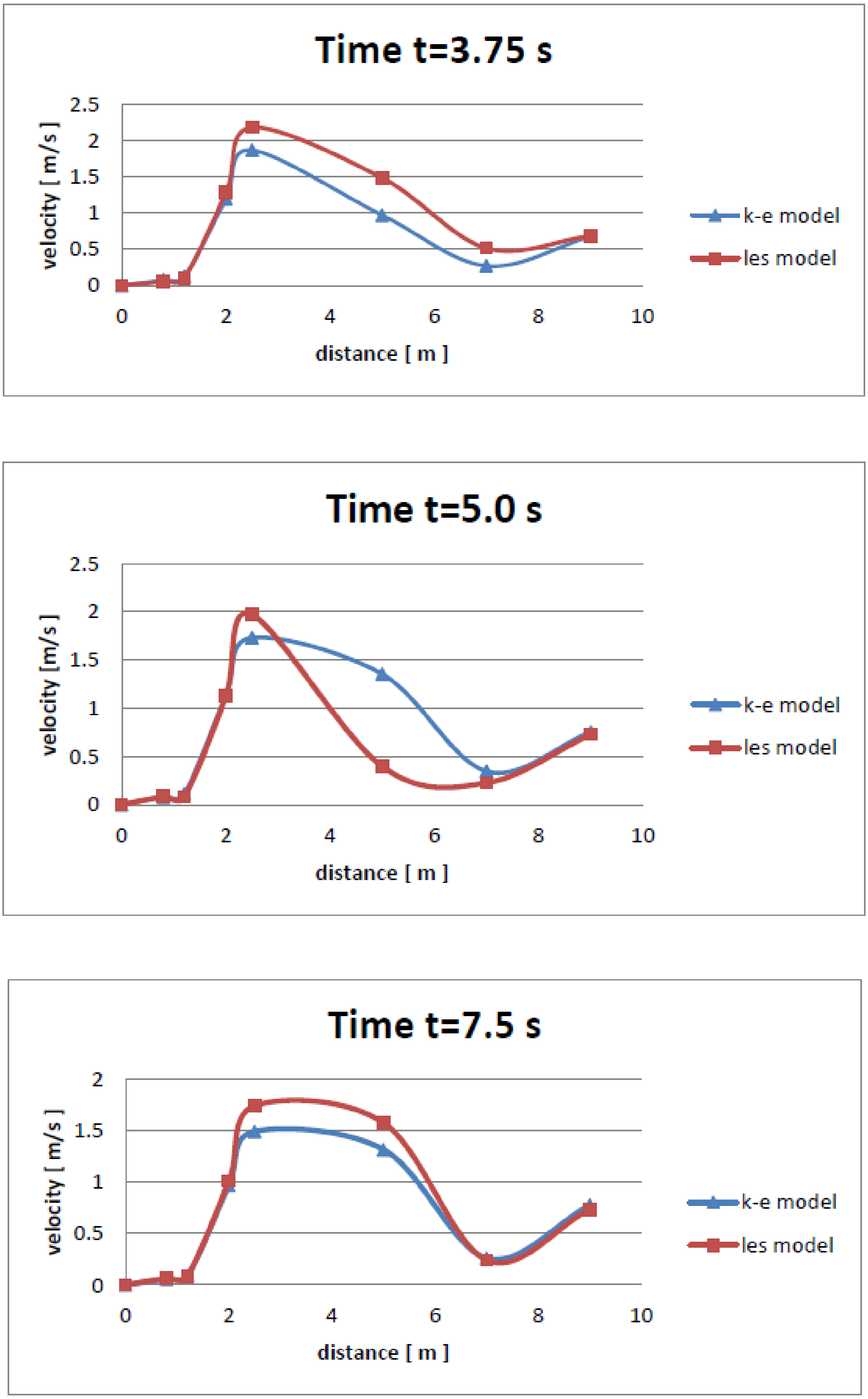


Figure 5.11: Variation of velocity with the length of dam at t= 3.75 s, 5 s &7.5 s.

The simulation was carried out with both the models like LES turbulence model and k-e model and observed that the LES model gave the peak velocity and downstream velocities higher than the k- epsilon model.

5.3.3 WATER SURFACE AND VELOCITY PROFILES

Extra correlations of results got with LES and k –e models are made by plotting water surface and velocity profiles at a given time.

The water surfaces along the length of the dam were tracked parallel to the Z axis at X= 0.15 m (near the wall), 1 m dam (center line), 1.25 m (near the edge of the gate) at time t= 2.5 s after the gate opening. The discontinuity of water surface was observed by the both LES and K-epsilon model at three different places parallel to Z axis.

The figure 5.12 shows the w- velocity (stream wise velocity), and figure 5.13 shows u- velocity (lateral velocity) profiles for positions of A, 11, 14 at time t= 1.25 s. the velocity profiles shows that the magnitude of the velocity obtained in the LES model was higher than the K-e model. At the position A the w- velocity increases with increase in depth and reaches higher value then starts decreases. The u-velocity (lateral velocity) was zero at the position A which was at the center line of dam, on this line the flow is symmetric. Areas 11 and 14 are found downstream of the dam quickly outside of the gate. At both areas, the velocity anticipated by the LES model is fundamentally higher. Note that the velocity profiles at areas 11 and 14 are mirror picture of each other because of the stream symmetry. The distinction in anticipated velocity profiles by the LES and k −e model highlights the limitation of the k −e model in predicting highly transient flow. The w-velocity and u-velocities were shown in below figure.

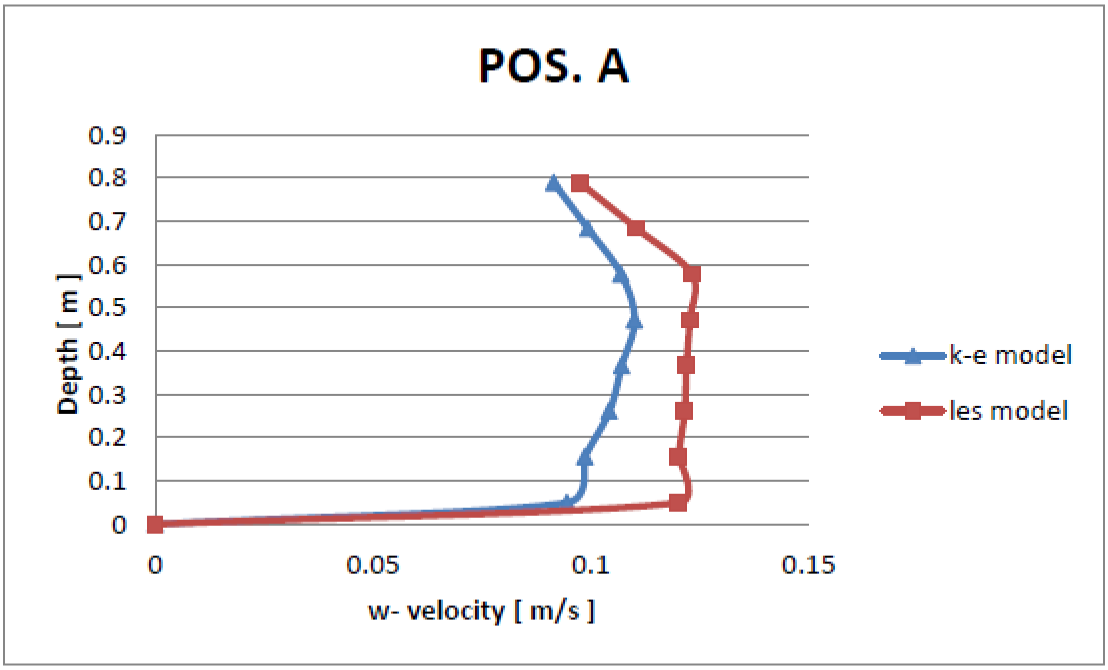


Figure 5.12: Comparison of w- velocity profiles (stream wise velocity), at location A using LES and k-e model at t=1.25 s.

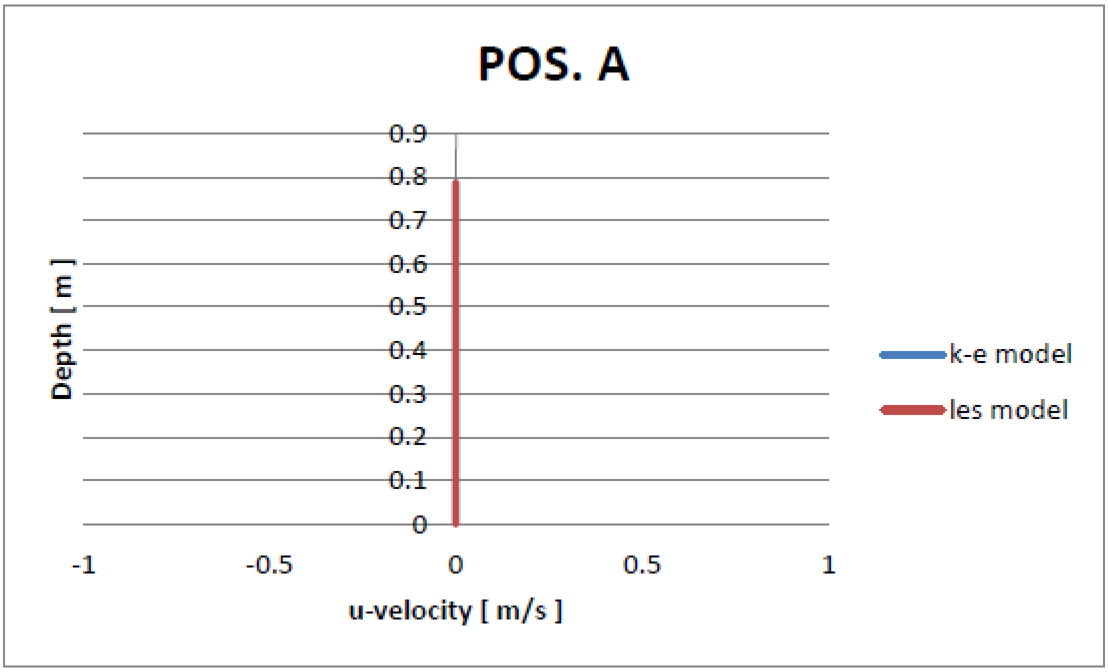


Figure 5.13: Comparison of u- velocity profiles (lateral velocity), at location A using LES and k-e model at t=1.25 s.

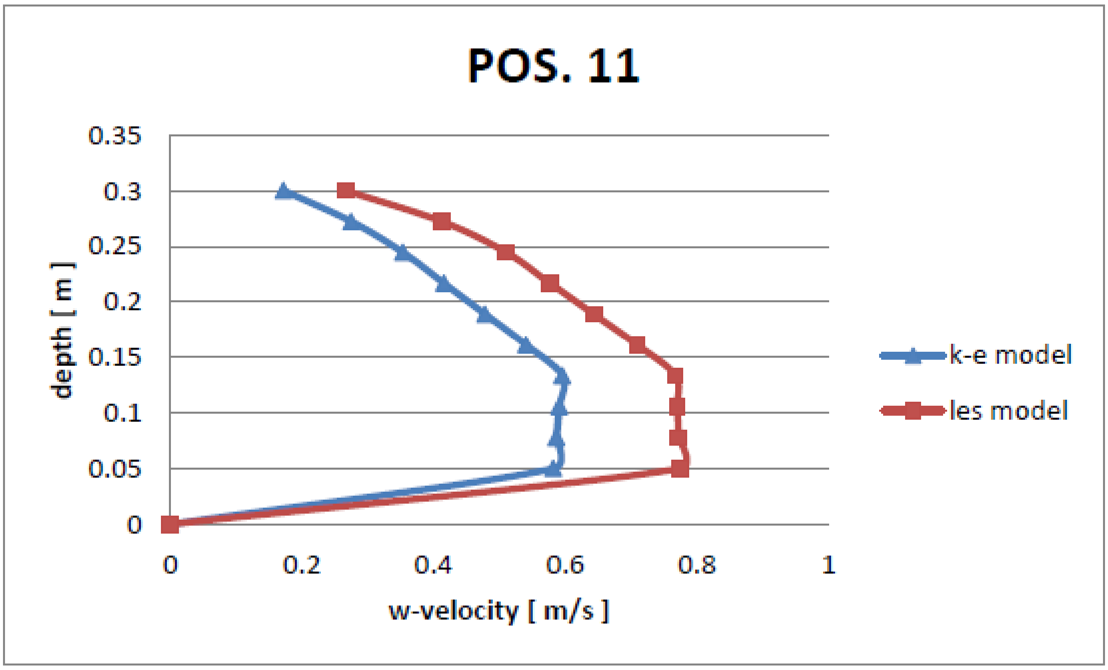


Figure 5.14: Comparison of w- velocity profiles (stream wise velocity), at location 11 using LES and k-e model at t=1.25 s.

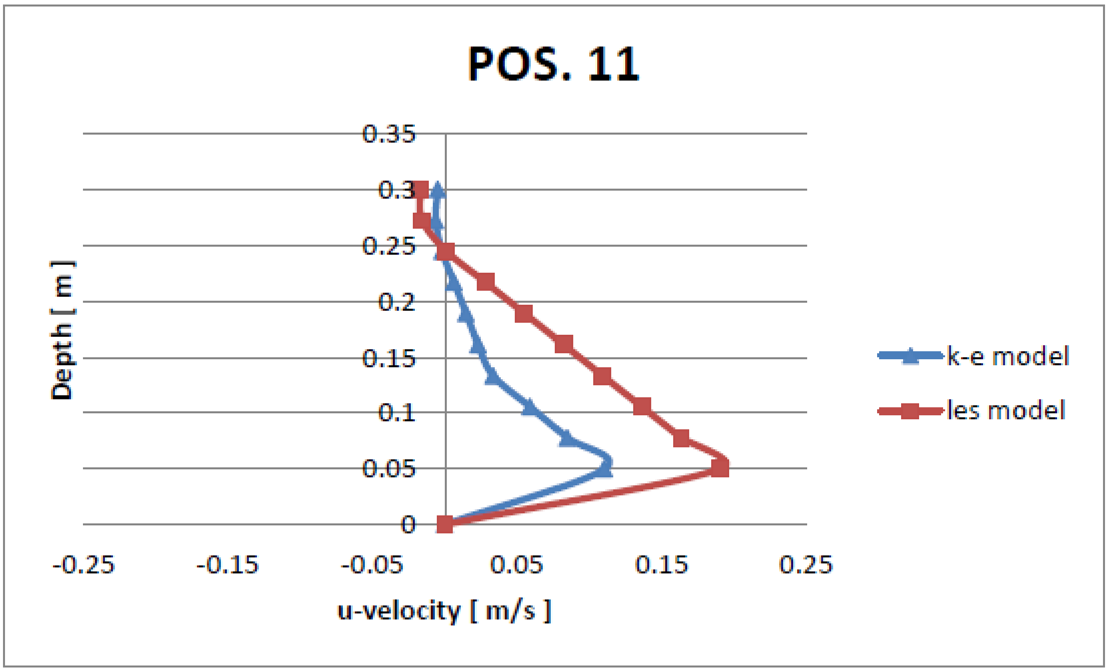


Figure 5.15: Comparison of u- velocity profiles (lateral velocity), at location 11 using LES and k-e model at t=1.25 s.

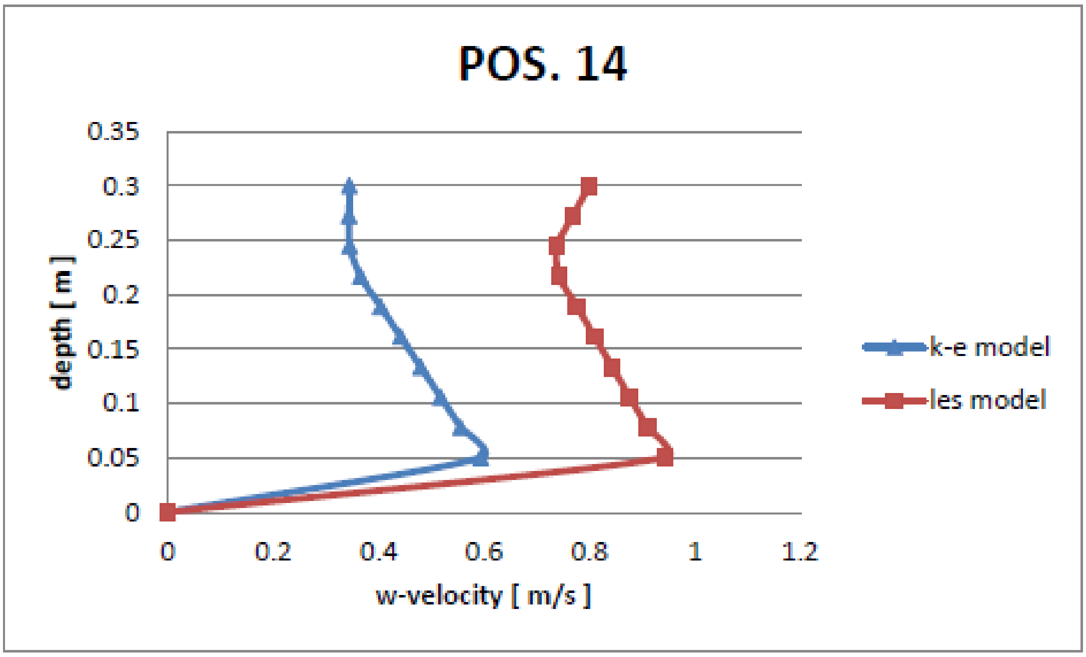


Figure 5.16: Comparison of w- velocity profiles (stream wise velocity), at location 14 using LES and k-e model at t=1.25 s.

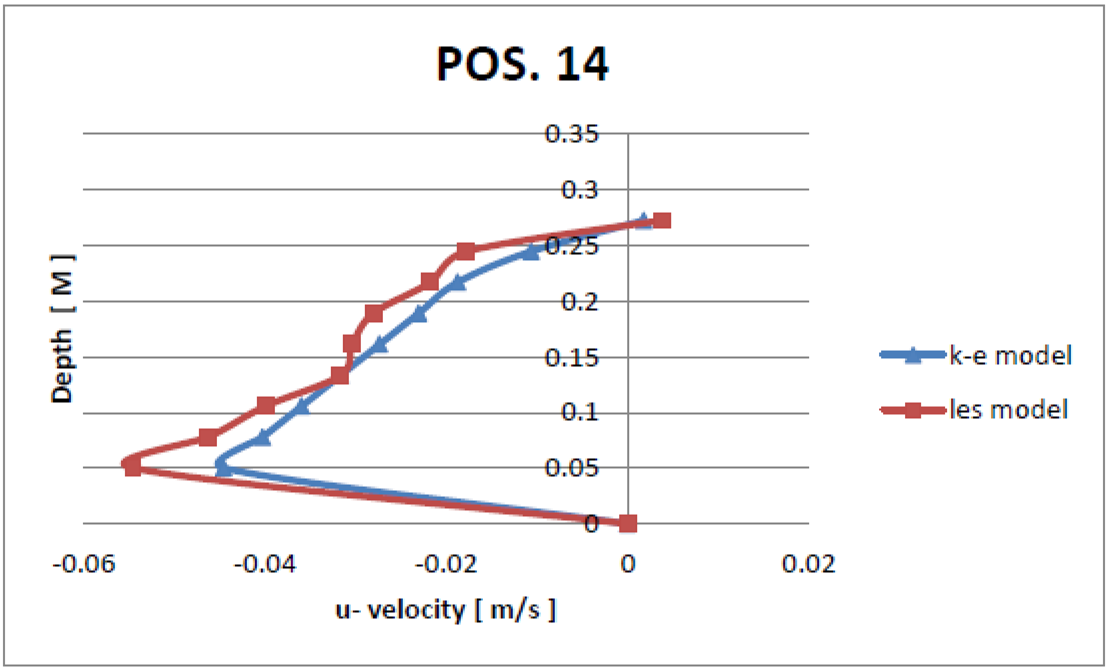
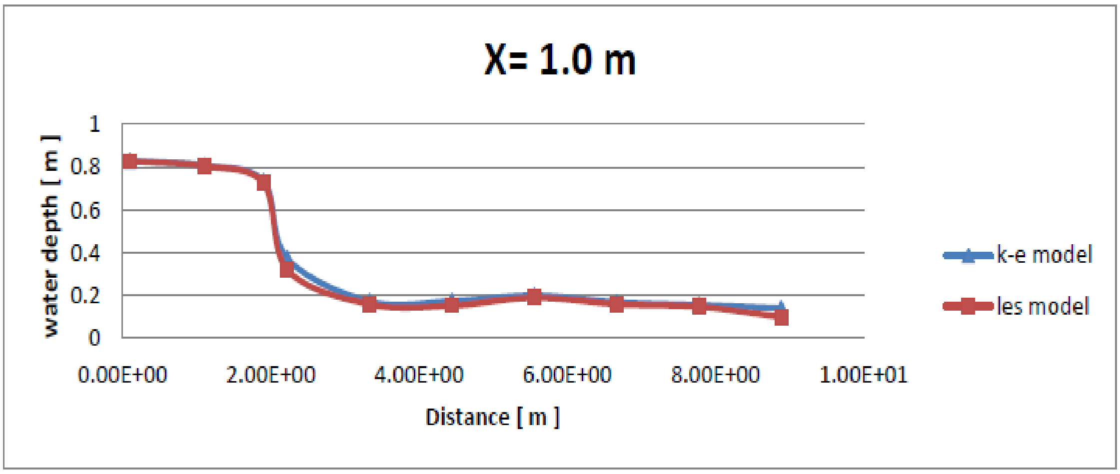
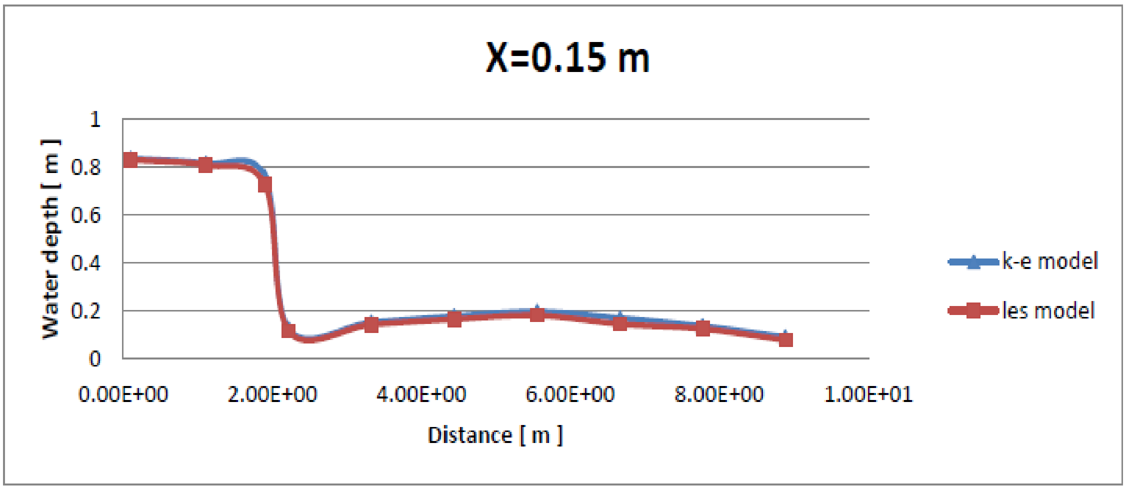


Figure 5.17: Comparison of u- velocity profiles (lateral velocity), at location 14 using LES and k-e model at t=1.25 s.





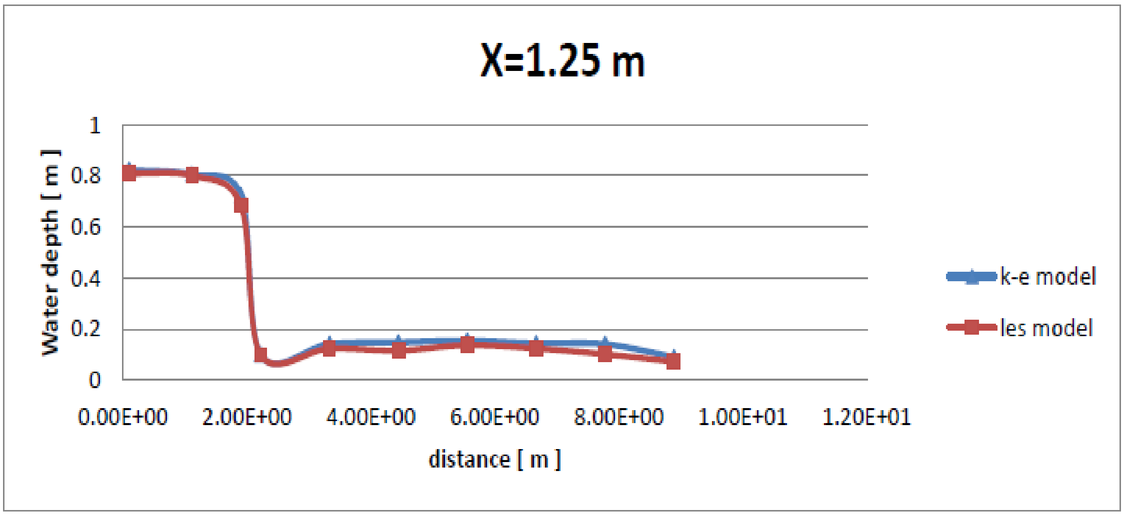
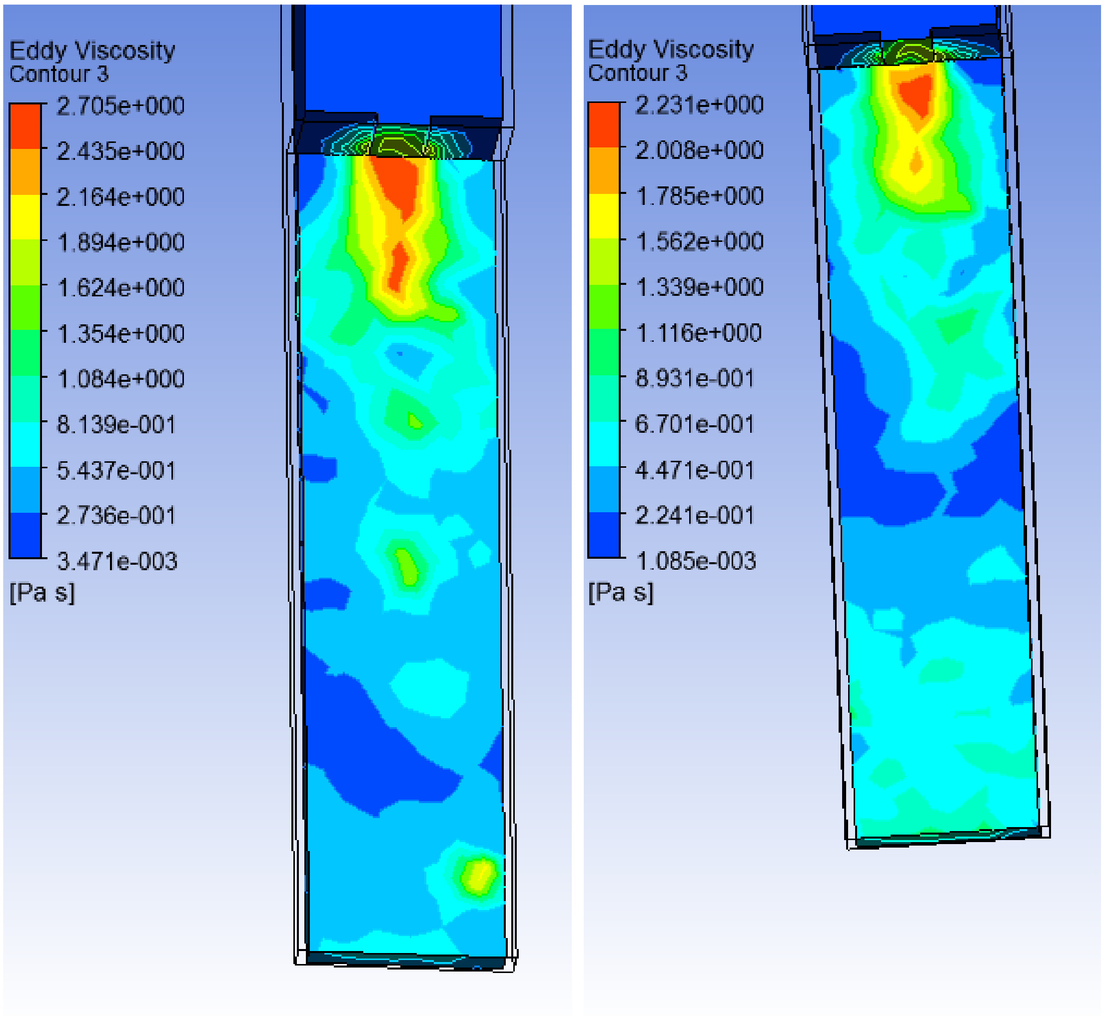


Figure 5.18: Variation of water depths with the length of the dam at different locations at t =1.25s.

5.3.4 EDDY VISCOSITY

The exchange of energy created by turbulent eddies is regularly demonstrated with a successful eddy viscosity likewise as the momentum exchange brought about by molecular diffusion is demonstrated with a molecular viscosity. The hypotheses that the impact of turbulent whirlpools on the stream can be demonstrated in this are regularly referred to as the Boussinesq eddy viscosity supposition and it was initially formulated by Boussinesq in 1877. The eddy viscosity is likewise ordinarily called the turbulent viscosity. The variation of eddy viscosity contours were shown figure 5.15.

At time t=2.5 sec At time t=5 sec



At time t=5 sec

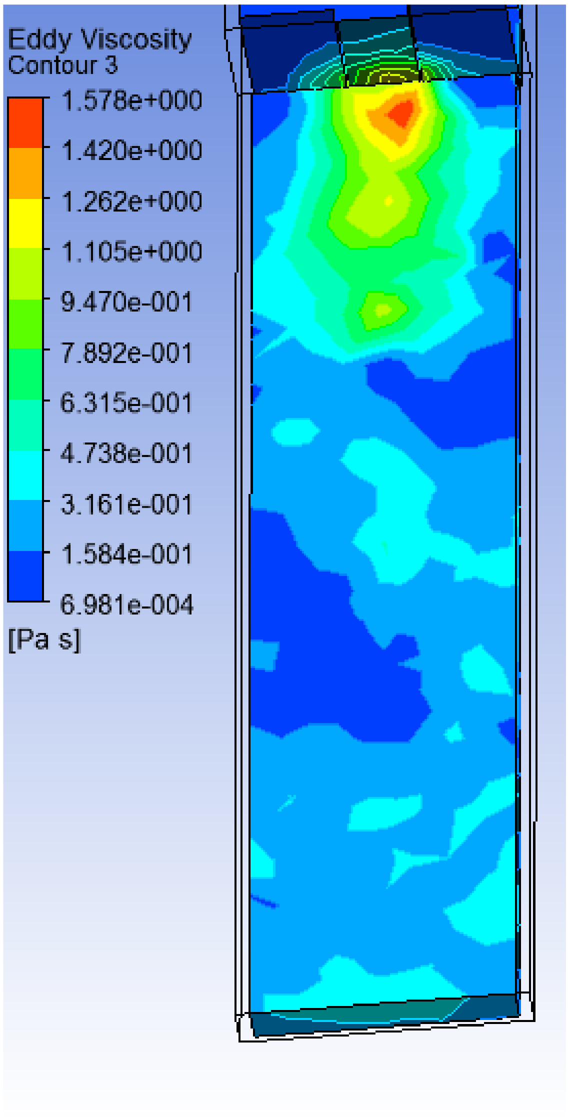


Figure 5.19: Variation of eddy viscosity with time.

The eddy viscosity was more at time t = 2.5 s, which means the transfer of momentum caused by turbulent eddies was more. For other times like t = 5.0 s and t = 7.5 s the eddy viscosity was reduced which shows that the turbulence at the downstream location is decreasing with the time.

CHAPTER 6

CONCLUSIONS

6.1 OVERVIEW

The Dam break simulation was carried out in ANSYS FLUENT using the large eddy simulation turbulence model and k-epsilon model, water surface was tracked by utilizing the Volume of Fluid method. Both the methods were compared to each other.

6.2 CONCLUSIONS

Three-dimensional numerical modelling of dam-break stream has been directed utilizing both LES and $k - e$ turbulence models and a VOF approach for surface following. The outcomes exhibit that 3-D numerical models, especially with LES turbulence closure, can give dependable and nitty gritty after effects of the stream. The results clearly demonstrate that both 3-D effects and turbulence are important in dam-break flows. Be that as it may, utilization of 3-D model for field-scale re-enactment will be computationally costly. A sensible solution might utilize 3-D models in the close field and SWE models in the far field areas. The findings in this thesis can be summarised as follows

- Velocity profiles predicted with the LES and k -epsilon turbulence model show significant differences especially in the vicinity of the dam downstream of the opening.

- It was observed that the LES model gave the peak velocity at downstream location higher than that of the k- epsilon model.

- The bottom pressure predicted with the LES and k- epsilon turbulence model show the similarity in the upstream area and significant variations occur in the downstream of the dam.

- The LES model successfully captures the fluctuations in temporal variation of water depth and velocity. At the center line the water depth varies similarly in both the LES and k-e model, but some differences shown along near wall region and parallel to the edge of the gate.

- The numerical models predict a faster rise of the peak velocity as compared to the observed data. The most likely reason for this difference is the instantaneous gate opening in the model.

- From the simulation it is clear that eddy viscosity is maximum near the gate opening which means turbulence is more and it reduces along the downstream side. Also the turbulence is decreasing in the downstream section with time.

6.3 SCOPE FOR FUTURE WORK

There is a lot of scope for the work to be done in future in this study area i.e. 3-D dam break flow analysis using the ANSYS.

Future scope for the present work was summarized as below:

- The physical modelling on the dam break flow have to be done and flow analysis for different parameters like velocity profiles, water surface elevation, bottom pressure.

- Wet bed conditions can also be used in ANSYS along with dry bed, as in only dry bed was used in the present research.

- Results can be compared from both experimental and software simulation obtained from ANSYS.

REFERENCES

Abdel-Sattar, A. M., A. A. Kassem, and M. H. Chaudhry (2008). “Case study: 17th street canal breach closure procedures”. Journal of Hydraulic Engineering 134(11), 1547–1558.

Adrian, R. J. (1991). “Particle-Imaging techniques for experimental fluid mechanics” .*Annual* Review of Fluid Mechanics 23(1), 261–304.

Akanbi, A. A. and N. Katopodes(1988). “Model for flooding propagation on initially dry land”.Journal of Hydraulic Engineering, 114(7),689–706.

Alam, M. M. and M. A. Bhuiyan (1995). “Collocation Finite-Element simulation of Dam-Break flows”. Journal of Hydraulic Engineering 121 (2), 118.

Alcrudo, F. and P. Garcia-Navarro (1993). “A high-resultiongodunov-type scheme in finite volumes for the 2d shallow water equations”. International Journal for Numerical Methods in Fluids *16* (6), 489–505.

Aleixo, R., S. Soares-Frazio and Y. Zech (2011). “Velocity-field measurements in a dam-break flow using a PTV imaging technique”. Experiments inFluids 50(6), 1633–1649.

Aleixo, R., B. Spinewine, S. S. Frazao, and Y. Zech (2011). “Velocity profiles in dam-break flows”: Water and sediment layers. River Flow 2011.

Aureli, F., A. Maranzoni, P. Mignosa, and C. Ziveri (2008). “Dam-Break flows: Acquisition of experimental data through an imaging technique and 2D numerical modelling”. Journal of Hydraulic Engineering 134(8), 1089.

Aziz, M.(2000). “Dam breaching model”. In Water Resources 2000, Minneapolis, Minnesota, USA, pp. 56–56.

Baskaran, A. and T. Stathopoulos (1989). “Computational evaluation of wind effects on buildings”. Building and Environment 24(4), 325–333.

Denlinger, R. and D. O’Connell (2008). “Simulations of cataclysmic outburst floods from pleistocene glacial Missoula”.Geological Society of America Bulletin(5-6), 678–689.

Dressler, R. (1952). "Hydraulic resistance effect upon the dam-break functions". Journal of Research, National Bureau of Standards 49(3).

E. Zedler, R. L. S. (2001). "Large-eddy simulation of sediment transport: Currents over ripples". Journal of Hydraulic Engineering 127(6), 444–452.

Eaket, J., F. Hicks, and A. Peterson (2005). "Use of stereoscopy for dam break flow measurement". Journal of Hydraulic Engineering 131(1), 24–29.

Elkholy, M. and M. H. Chaudhry (2009). "Tracking sandbags motion during levee breach closure using dptv technique". In 33rd Congress, International Association of Hydraulic Engineering and Research, IAHR, Vancouver, BC, Canada, pp. 5992–5999.

Elliot, R. and M. Chaudhry (1992). "A wave propogation model for two dimensional dam-break flows". Journal of Hydraulic Research 30, 467–483.

FEMA (1986)."*Design Manual for Retrofitting Flood-Prone Residential Structures"*, Volume No. 114. Federal Emergency Management Agency.

Fennema, R. and M. Chaudhry (1989). "Implicit method for two-dimensional unsteady free-surface flows".Journal of Hydraulic Research 27(3), 331–332.

Fennema, R. and M. Chaudhry (1990). "Explicit method for 2-d transient free-surface flows".Journal of Hydraulic Research 116(8), 1013–1034.

Ferziger, J. and M. Peric (2003). "Computational Methods for Fluid Dynamics"(3rd ed.). Springer. FLUENT (2006). Fluent 6.1 user manual. Fluent.

Fraccarollo, L. and E. Toro (1995). "Experimental and numerical assessment of the shallow water model for two-dimensional dam-break type problems".Journal of Hydraulic Research 363(6), 843–864.

Franco, G., R. Green, B. Khazai, A. Smyth, and G. Deodatis (2010). "Field damage survey of new orleans homes in the aftermath of hurricane katrina". NaturalHazards Review *11* (1), 7–18.

Golub, G. and C. van Loan (1996). "Matrix Computations". Johns Hopkins University Press.

Hartley, R. and S. Kang (2008). "Parameter free radial distortin correction with center of distortion estimation". IEEE Trans. *29*, 1309–1321.

He, P. (1997, jun). "Numerical simulation of air flow in an urban area with regularly aligned blocks". Journal of Wind Engineering and Industrial Aerodynamics 67-68, 281–291.

Hirt, C. and B. Nichols (1981). "Volume of fluid (vof) method for the dynamics of free boundaries".Journal of Computational Physics 39, 201–225.

Hunt, B. (1983). "Perturbation solution for Dam-Break floods".Journal of Hydraulic Engineering 110(8), 1058.

Hunt, B. (1984). "Dam-break solution".Journal of Hydraulic Engineering 110(6), 675–685. IPET (2009). "Performance evaluation of the new orleans and southeast Louisiana hurricane protection system". Final report voli-ix, US Army Corps of Engineers, Washington DC.

Janosi, I., D. Jan, K. G. Szabo, and T. Tel (2004). "Turbulent drag reduction in dam-break flows". Experiments in Fluids 37, 219–229.

Jha, A. K., J. Akiyama, and M. Ura (1995). "First- and Second-Order flux difference splitting schemes for Dam-Break problem". Journal of Hydraulic Engineering121(12), 877.

Kantoush, S., G. D. Cesare, and A. Schleiss (2007). "Flow field investigation in a rectangular shallow reservoir using uvp, lspiv and numerical modelling". Flow Measurement and Instrumentation.

Katopodes, N. and T. Strelkoff (1978). "Computing two-dimensional dam-break flood waves".Journal of Hyd. Div. ASCE, 1269–1288.

Kikura, H., Y. Takeda, T. Taishi, M. Aritomi, and M. Mori (1998). "Flow mapping using ultrasonic doppler method". 1998 ASME Fluids Engineering Division Summer Meeting.

Kunreuther, H. (2006, jan). "Disaster mitigation and insurance: Learning from Katrina". The ANNALS of the American Academy of Political and Social Science 604(1), 208–227.