FLOW MODELLING OF NON PRISMATIC COMPOUND CHANNEL USING ANSYS

A DISSERTATION

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

MASTER OF TECHNOLOGY

IN

HYDRAULICS AND WATER RESOURCE ENGINEERING

Submitted By

SHAILESH JAISWAL

(Roll No. 2K16/HFE/16)

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, Shailesh Jaiswal, Roll No. 2K16/HFE/16 of M.Tech. Hydraulics and Water Resource Engineering, hereby declare that the project Dissertation titled **"Flow Modelling of Non Prismatic Compound Channel 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 **Shailesh Jaiswal**

Date:

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

CERTIFICATE

I hereby certify that the Project Dissertation titled **“Flow Modelling of Non Prismatic Compound Channel Using Ansys”** by **Shailesh Jaiswal**, Roll No. **2K16/HFE/16**, 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

Delhi Technological University
Delhi, INDIA

ACKNOWLEDGEMENT

First, I would like to thank **my family**. Without their love and support over the years, none of this would have been possible. They have always been there for me and I am thankful for everything they have helped me achieve.

I wish to express my deep sense of gratitude and veneration to my supervisor, **Prof. (Dr.) Bharat Jhamnani**, Department of Civil Engineering, Delhi Technological University, Delhi, for his perpetual encouragement, constant guidance, valuable suggestions and continued motivation, which has enabled me to complete this work.

I am also thankful to all the faculty members for their constant guidance and facilities to carry out my work.

Finally, I would like to thank my dear friends **Harshit Kumar Jayant, Dolon Bannerjee, Punj Pratap Singh and Abhishek.** For coming to lab daily, and staying up till I leave, we shared the same story and worked hard together. This experience would not have been fun without them.

SHAILESH JAISWAL
M.Tech (HFE), DTU
Roll NO. 2K16/HFE/16

Abstract

FLOW MODELLING OF NON PRISMATIC COMPOUND CHANNEL USING ANSYS

SHAILESH JAISWAL
DEPARTMENT OF CIVIL ENGINEERING
DELHI TECHNOLOGICAL UNIVERSITY
SUPERVISOR: PROF. (DR.) BHARAT JHAMNANI

Flooding situation in river is a complex phenomenon and affects the livelihood and economic condition of the region. The modeling of such flow is primary importance for a river engineers and scientists working in this field. Water surface prediction is an important task in flood risk management. As a result of topography changes along the open channels, designing the converging compound channel is an essential. Fluvial flows are strongly influenced by geometry complexity and large overall uncertainty on every single measurable property, such as velocity distribution on different sectional parameters like width ratio, aspect ratio and hydraulic parameter such as relative depth. The geometry selected for this study is that of a prismatic compound channel having converging flood plain. For the research work the parameters, the water depth, incoming discharge of the main channel and floodplains were varied. This total topic represents a practical method to predict lateral depth-averaged velocity distribution in a prismatic compound channel. Compound section of a natural channel generally comprises a wider and rougher flood plain than main channel. The flow process in the open channel becomes more complicated at over bank stages due to the different hydraulic condition prevailing in the main channel and the adjoining flood plain. As the shallow flood plains offer more resistance to flow than the deep main channel, the velocity tends to be higher in deeper main channel than the shallow flood plain. This variation of velocity between deep main channel section and the adjoining shallow flood plains raise the lateral momentum transfer, which further complicates the flow process. In the present work an experiment for the depth average velocity at different points of the channel cross section in lateral direction is carried out by using ADV, for a compound channel having width ratio 2.923

with differential roughness (the ratio of base n value of flood plain surface roughness to main channel roughness) 2.0833m^(-1/3)sec. The numerical model using ANSYS fluent as a result of developing simulation model for velocity and flow depth are compared with laboratory data for flow in a compound channel that consists of a main channel and symmetric flood plains set at a fixed bed slope. Reasonable agreement between the numerical results and experimental data is shown for steady uniform flow located at a section at a distance of 8.5m from the start end.

Computational Fluid Dynamics (CFD) is often used to predict flow structures in developing areas of a flow field for the determination of velocity, pressure, shear stresses, effect of turbulence and others. This study aims to validate CFD simulations of free surface flow or open channel flow by using Finite Volume method by comparing the data of the past research done in NIT Rourkela by K.K.Khatua.

CONTENTS

List of figures

Abbreviation

CFD - Computational Fluid Dynamics
FVM - Finite Volume Method
HOL - Height of Liquid
LES - Large Eddy Simulation
RNG - Renormalization Group
RSM - Reynolds Stress Model
VOF - Volume of fluid
Upred - Predicted velocity
Uobsd - Observed velocity
Vmean - Apparent channel Velocity
B - Width of the Channel
L - Length of the Channel
Q - Discharge of the Channel
R - Hydraulic Radius of the Channel
S - Slope of the Channel
H - Height of the Channel
y - Depth of water in a Channel

1

INTRODUCTION

1.1 BACKGROUND

An open channel is a passage in which liquid flows with a free surface .In other words the pressure is impressed on free surface is atmospheric. An open channel can be natural or artificial. Depending upon the shape, a channel is either prismatic or non-prismatic. A channel is said to be prismatic when the cross section is uniform and the bed slope is constant. Eg. Rectangular, trapezoidal, circular, parabolic. A channel is said to be non-prismatic when its cross section and slope changes. Ex: River & Stream. It is seen that, the river generally exhibit a two stage geometry (deeper main channel and shallow floodplain called compound section) having either prismatic or non-prismatic geometry (geometry changes longitudinally).

A compound section of a natural channel generally comprises a wider and rougher Floodplain than the main channel. The flow process in the open channel becomes more complicated at overbank stages due to the different hydraulic conditions prevailing in the main channel and the adjoining floodplains. For overbank stage, the resulting velocity distribution is generally not uniform across the cross-section; in particular the velocity tends to be higher in deeper main channel than the shallower floodplain, as in these compound channels the shallow floodplains offer more resistance to flow than the deep main channel. The velocity variation raise lateral momentum transfer between the deep main channel section and the adjoining shallow floodplains, which further complicates the flow process, leading to the uneven distribution of flow and shear stress in the main channel and floodplain regions.

In prismatic compound channels with roughed floodplains the resulting interactions and momentum exchanges is increased. This extra momentum exchange is very important parameter and should be taken into account in the overall flow modeling of a river. So research is still underway to develop methods which are physics based, have universal applicability and simple to apply. Because of the practical difficulty in obtaining sufficiently accurate and comprehensive field measurements of velocity and shear stress in compound channels under unsteady flood flow conditions well designed laboratory investigations under steady flow conditions are still preferred as a trusted method to provide the information concerning the details of the flow structure. Such information is important in the application and development of numerical models aimed at solving certain practical hydraulic problems (i.e. to understand the mechanism of sediment transport, analysis of river migration, to prevent bank erosion in river channel, design stable channels, flood risk management, etc.). Knight and Hammed (1984) extended the work of Knight and Demetrious (1983), to the compound channels having rough floodplains. By adding roughness elements, the floodplains were roughened. They studied the influence of differential roughness between floodplain and main channel on the process of lateral momentum transfer using dimensionless channel parameters (e.g. the width ratio, depth ratio, Roughness ratio and aspect ratio). Myers et al. (2001) presented of an experimental results of a compound channel having fixed and mobile main channel along with two rough floodplains. They investigated velocity and discharge relationships illustrating the complex behavior of compound channel river section. Hin and Bessaih (2004) investigated velocity distribution, stage-discharge relationship and the effect of momentum transfer in a straight compound channel having a rougher floodplain than the main channel. They artificially roughened the floodplain by using wire mesh. Seckin (2004) investigated the reliability and performance of four different one dimensional methods of computing the discharge capacity for compound channels by conducting a series of experiments in a compound channel having a smooth main channel and smooth or rough floodplains.

For the study, he roughened the floodplains in four different ways using metal meshes. The metal meshes had a width of35.5cm, a height of 14.5cm and an angle of 30^o and were placed at 4 different intervals spacing on each floodplain in order to provide a particular roughness. A separate series of experiments were undertaken to find out their exact resistance properties of floodplain roughness were created. Most experimental efforts have been concentrated on homogeneous roughness (smooth) compound channels constrained to low width ratio (width of compound channel / width of main channel base).Therefore, the present study intend to obtain information about the influencing capacity of differential roughness on flow structure such as depth-averaged velocity, in an idealised compound section having width ratio(α)=3.

1.2 RIVER SYSTEM

Rivers play an integral part in the day to day functioning of our planet. Therefore it is important to understand the flow characteristics of rivers in both their in bank and overbank flow condition. An open channel is a passage in which liquid flows with a free surface. An open channel is a passage in which liquid flows with a free surface, open channel flow has uniform atmospheric pressure exerted on its surface and is produced under the action of fluid weight. It is more difficult to analyze open channel flow due to its free surface. Flow is an open channel is essentially governed by Gravity force apart from inertia and viscous forces.

Examples of Open Channel Flow

- The natural drainage of water through the numerous creek and river systems.
- The flow of rainwater in the gutters of our houses.
- The flow in canals, drainage ditches, sewers, and gutters along roads.

- The flow of small rivulets, and sheets of water across fields or parking lots.
- The flow in the chutes of water rides.

An open channel is classified as ***natural*** or ***artificial***.

Natural: Open channels are said to be natural when channels are irregular in shape, alignment and surface roughness. Eg. Streams, rivers, estuaries etc.

Artificial: When the open channels are regular in shape, alignment and uniform surface roughness which are built for some specific purpose, such as irrigation, water supply, water power development etc. are called as artificial open channels.

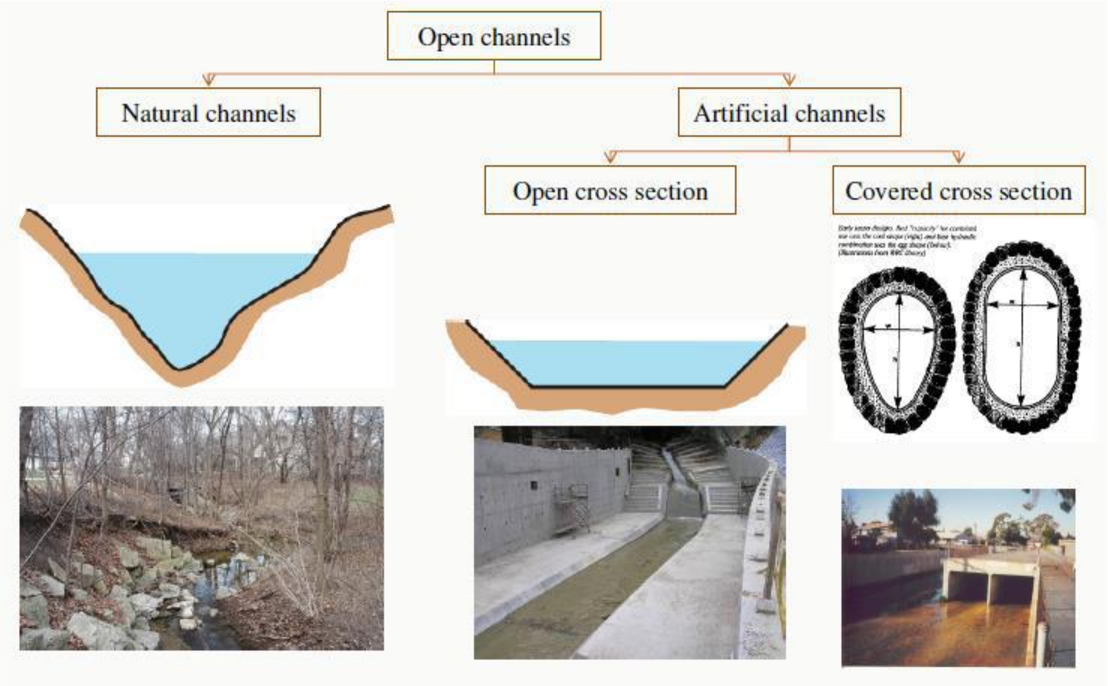


Figure 1.1. Classification of Open channels

The river generally exhibit a two stage geometry :

- Deeper main channel.
- Shallow floodplain called compound section.

Compound Channel:

When the flow is out of bank, like during flood it is known as compound channel flow. Generally compound
channels are classified as

- Prismatic compound channel
- Non prismatic compound channel

Prismatic compound channels:

A channel is said to be prismatic when the cross section is uniform and the bed slope is constant and having fixed alignment.
Eg. Most of the manmade channels i.e. Rectangular, trapezoidal, circular and parabolic.

Non prismatic compound channels:

A channel is said to be non-prismatic when its cross sectional shape, size and bottom slope are not constant longitudinally. Eg. All the natural channels i.e. River, Streams and Estuary. Some examples of non-prismatic channels are flow through culverts, flow through bridge piers, high flow through bridge pier and obstruction, channel junction etc. Non-prismatic compound channel may be converging and diverging or skewed type.

1.3 RIVER AND FLOODING

An open channel is a passage in which liquid flows with a free surface .In other words the pressure is impressed on free surface is atmospheric. An open channel can be natural or artificial. Depending upon the shape, a channel is either prismatic or non-prismatic. A channel is said to be prismatic when the cross section is uniform and the bed slope is constant. Ex. Rectangular, trapezoidal, circular, parabolic.

A channel is said to be non-prismatic when its cross section and slope changes. Ex: River &Stream. It is seen that, the river generally exhibit a two stage geometry (deeper main channel and shallow floodplain called compound section) having either prismatic or non-prismatic geometry (geometry changes longitudinally).A compound section of a natural channel generally comprises a wider and rougher Floodplain than the main channel. The flow process in the open channel becomes more Complicated at overbank stages due to the different hydraulic conditions prevailing in the main channel and the adjoining floodplains. For overbank stage, the resulting velocity distribution is generally not uniform across the cross-section; in particular the velocity tends to be higher in deeper main channel than the shallower flood plain, as in these compound channels the shallow floodplains offer more resistance to flow than the deep main channel. The velocity variation raise lateral momentum transfer between the deep main channel section and the adjoining shallow floodplains, which further complicates the flow process, leading to the uneven distribution of flow and shear stress in the main Channel and flood plain regions.

In prismatic compound channels with roughed floodplains the resulting interactions and momentum exchanges is increased. This extra momentum exchange is very important parameter and should be taken into account in the overall flow modeling of a river. So research is still underway to develop methods which are physics based, have universal applicability and simple to apply. Because of the practical difficulty in obtaining sufficiently accurate and comprehensive field measurements of velocity and shear stress in compound channels under unsteady flood flow conditions well designed laboratory investigations under steady flow conditions are still preferred as a trusted method to provide the information concerning the details of the flow structure. Such information is important in the application and development of numerical models aimed at solving certain practical hydraulic problems (i.e. to understand the mechanism of sediment transport, analysis of river migration, to prevent bank erosion in river channel, design stable channels, flood risk management, etc.).**Knight** and **Hammed** (1984) extended the work of Knight and Demetrious (1983), to the compound channels having rough floodplains. By adding roughness elements, the floodplains were roughened. They studied the influence of differential roughness between floodplain and main channel on the process of lateral momentum transfer using dimensionless channel parameters (e.g. the width ratio, depth ratio, Roughness ratio and aspect ratio).

Myers et al. (2001) presented an experimental results of a compound channel having fixed and mobile main channel along with two rough floodplains. They investigated velocity and discharge relationships illustrating the complex behaviour of compound channel river section. Hin and Bessaih (2004) investigated velocity distribution, stage-discharge relationship and the effect of momentum transfer in a straight compound channel having a rougher floodplain than the main channel. They artificially roughened the floodplain by using wire mesh. Seckin (2004) investigated the reliability and performance of four different one dimensional methods of computing the discharge capacity for compound channels by conducting a series of experiments in a compound channel having a smooth main channel and smooth or rough floodplains. For the study, he roughened the floodplains in four different ways using metal meshes. The metal meshes had a width of35.5cm, a height of 14.5cm and an angle of 30^o and were placed at 4 different intervals spacing on each floodplain in order to provide a particular roughness. A separate series of experiments were undertaken to find out their exact resistance properties of floodplain roughness were created. Most experimental efforts have been concentrated on homogeneous roughness (smooth) compound channels constrained to low width ratio (width of compound channel / width of main channel base).Therefore, the present study intend to obtain information about the influencing capacity of differential roughness on flow structure such as depth-averaged velocity, in an idealized compound section having width ratio(α)=3.

1.4 NUMERICAL MODELLING

Computational Fluid Dynamics (CFD) is a computer based numerical analysis tool. The growing interest on the use of CFD based simulation by researchers have been identified in various fields of engineering as numerical hydraulic models can significantly reduce costs associated with the experimental models. The basic principle in the application of CFD is to analyze fluid flow in-detail by solving a system of non-linear governing equations over the region of interest, after applying specified boundary conditions. A step has been taken to do numerical analysis on a non-prismatic compound channel flow having converging floodplains. The work will help to simulate the different flow variables in such type of complex flow geometry. The use of computational fluid dynamics was another integral component for the completion of this project since it was the main tool of simulation.

In general, CFD is a means to accurately predict phenomena in applications such as fluid flow, heat transfer, mass transfer, and chemical reactions. There are a variety of CFD programs available that possess capabilities for modeling multiphase flow. Some common programs include ANSYS and COMSOL, which are both multi physics modeling software packages and FLUENT, which is a fluid-flow-specific software package. 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. CFD is also useful for design since it can be directly translated to a physical setup and is cost-effective (Bakker et al., 2001).

In the present work, an effort has been made to investigate the velocity profiles for five different sections of a compound channel having converging flood plain by using a computational fluid dynamics (CFD) modeling tool, named as FLUENT. The CFD model developed for a real open-channel was first validated by comparing the velocity profile obtained by the numerical simulation with the actual measurement carried out by experimentation in the same channel using Preston tube. The CFD model has been the used to analyze the effects of flow due to convergence of flood plain width and bed slope, and to study the variations in velocity profiles along the horizontal and vertical directions. The simulated flow field in each case is compared with corresponding laboratory measurements of velocity and water surface elevation. Computational Fluid Dynamics (CFD) is a mathematical tool which is used to model open channel ranging from in-bank to over-bank flows. Different models are used to solve Navier-Stokes equations which are the governing equation for any fluid flow. Finite volume method is applied to discretize the governing equations. The accuracy of computational results mainly depends on the mesh quality and the model used to simulate the flow.

1.5 ADVANTAGES OF NUMERICAL MODELLING

Despite exact results and clear understanding on flow phenomena, experimental approach has some drawbacks such as laborious data collection and data can be collected for limited number of points due to instrument operation limitations; the model is usually not at full scale and the three dimensional flow behavior or some complex turbulent structure which is the nature of any open channel flow cannot effectively captured through experiments. So in these circumstances, computational approach can be adopted to overcome some of these issues and thus provide a complementary tool. In comparison to experimental studies; computational approach is repeatable, can simulate at full scale; can generate the flow taking all the data points into consideration & moreover can take greatest technical challenge *i.e.;* prediction of turbulence. The complex turbulent structures like secondary flow cells, vortices, Reynolds stresses can be identified by numerical modeling effectively which are quite essential for the study of energy outflow in open channel flows. Many researchers in the recent centuries have numerically modeled open channel flows and has successfully validated with the experimental results.

2

LITERATURE REVIEW

2.1 OVERVIEW

This chapter outlines about the previous research done by other researchers in the field of open channel flow which is relevant to the current work. Distribution of flow velocity in longitudinal and lateral direction is one of the important aspects in open channel flows. It directly relates to several flow features like water profile estimation, shear stress distribution, secondary flow and channel conveyance.

The distribution of velocity in open channel flow is generally affected by various factors such as channel geometry, types of channel and patterns of channel, channel roughness and sediment concentration in flow which have critically studied by many renowned researchers. Many approaches are there for predicting stage discharge relationships, velocity distribution and boundary shear distribution on main channel and flood plain perimeter which are mainly applicable to prismatic compound channels. There are many study found in literature related to both prismatic and non-prismatic compound channels flow.

Previous Experimental Research on Prismatic Compound Channel

Many practical problems in river engineering require accurate prediction of flow in compound channels. Over-bank channels can be characterized by a deep main channel, bounded on one or both sides by a relatively shallow floodplain, which is often hydraulically irregular. Consequently, velocities in the main channel tend to be significantly greater than those on the floodplain. This difference in velocity can lead to large velocity gradients in the region of the interface between the main channel and floodplains. Likewise, local flow conditions determine the erosion and deposition rates of sediment in the main channel and floodplains.
Therefore, accurate prediction of discharge capacity of compound channels is essential for flood mitigation systems. The flow structure in open channels is highly dependent on the regime of the flow, i.e. laminar or turbulent. It also depends upon existence of vortices at various length scales, acting toward all three directions, typically generated by high shear between fluid layers and its boundaries, as noticed by ***W.D. KNIGHT , M.E. HAMED (1984)***. Such vortices are a form of energy transfer that converts part of the kinetic energy of the flow into heat through viscosity.

The common types of vortices that develop in open channel flow are due to surface roughness, the anisotropy of turbulent velocity fluctuations in the y and z directions, leading to secondary flows and high velocity gradients between the main channel and floodplain, explained by **BHOWMIK N. G., DEMISSIE M. (1982)** leading to plan form vortices at this interface. Each of these components is described clearly in the following sections.

By the measurement of stage–discharge relationship and observation of velocity fields in small laboratory two stage channels **K.K. KHATUA.,K.C. PATRA and P.K. MOHANTY (2012)** have found the zones of interaction between the main channel and floodplain flows which was occupied the whole or at least very large portion of the main channel. They have also explained that the water, which approaches the channel by way of floodplain, penetrates to its full depth and there is a dynamic exchange of water between the inner channel and floodplain. This lead to consequent circulation in the channel in the whole section. The energy dissipation mechanism of the trapezoidal section was found to be quite different from the rectangular section and they have suggested for further study in this respect. They have also suggested for further investigation to quantify the influence of flood plain roughness on flow parameters.

Khatua and Patra (2012a and 2012b) have presented apparent shear stress and developed a new method named as MDCM to predict the stage discharge relationships in compound channel of higher width ratio.

Brian M. Stone and Hung Tao Shen (2002) conducted experiments on the hydraulics of flow in an open channel with circular cylindrical roughness. The laboratory studies of an extensive set of flume experiments for flows with emergent and submerged cylindrical stems of various sizes and concentrations were done. The results shown that the flow resistance varies with flow depth, stem concentration, stem length, and stem diameter. The stem resistance experienced by the flow through the vegetation is best expressed in terms of the maximum depth-averaged velocity between the stems. Physically based formulas for flow resistance, the apparent channel velocity, and flow velocities in the roughness and surface layers are developed. The formulas are validated with the flume data from the present study as well as those from past studies. A method for calculating channel hydraulic conditions using these formulas is presented.

M. Righetii and A. Armanini (2002) reported that the values of roughness coefficient depend on degree of submergence. In the wet season with high discharge, vegetation undergoes high degree of bending resulting complete submergence in most cases. At that time, the values of the coefficient is less as compared to the values when the grasses are partly submerged in dry season having low flows.

Fu-Chun Wu, Hsieh Wen Shen and Yi-Ju Chou (1998) calculated the variation of the vegetative roughness coefficient with the depth of flow. A horsehair mattress is used in the experimental study to simulate the vegetation on the watercourses. Test results reveal that the roughness coefficient reduces with increasing depth under the unsubmerged condition. However, when fully submerged, the vegetative roughness coefficient tends to increase at low depths but then decrease to an asymptotic constant as the water level continues to rise. A simplified model based on force equilibrium is developed to evaluate the drag coefficient of the vegetal element; Manning's equation is then employed to convert the drag coefficient into the roughness coefficient. The data of this study are compared with those of selected previous laboratory and field tests. The results show a consistent trend of variation for the drag coefficient versus the Reynolds number.

3

METHODOLOGY

In order to find out the effect of diversity in floodplain roughness and main channel roughness on the flow characteristics (i.e. boundary shear distribution, flow distribution, depth-averaged velocity, variation in overall and zonal Manning's *n* and discharge) during over flow condition in a compound channel, experiments were conducted under controlled laboratory conditions in the Fluid Mechanics and Hydraulics Laboratory of the Civil Engineering Department at the Delhi Technological University, India.

SL NO	DESCRIPTION	EXPERIMENTAL CHANNEL
1	Channel type	Compound channel with rough flood plain
2	Flume size	15m×1.9m×0.125m
3	Geometry of main channel section	Trapezoidal(side slope1:1)
4	Channel width	65 cm at bottom and 90 cm at top
5	Bed slope of channel	1:1
6	Main channel height	0.125m
7	Relative depth	0.4
8	Flood plain roughness	Gravel(0.025m^(-1/3)sec)
9	Channel bed slope	0.0022

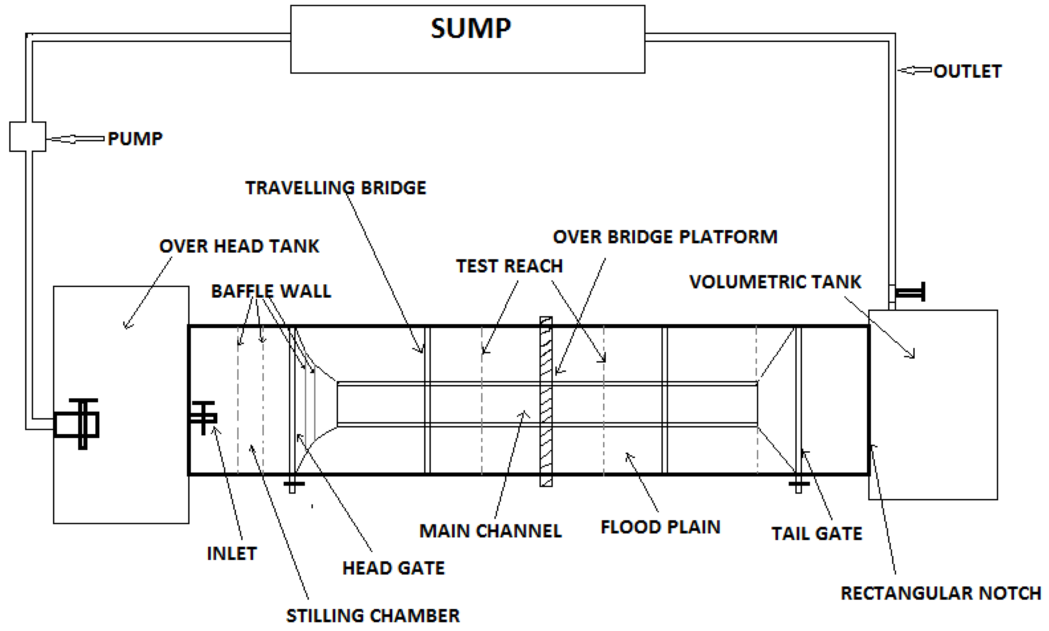


Fig 3.1 Plan View Of Experimental Setup Of The Channel

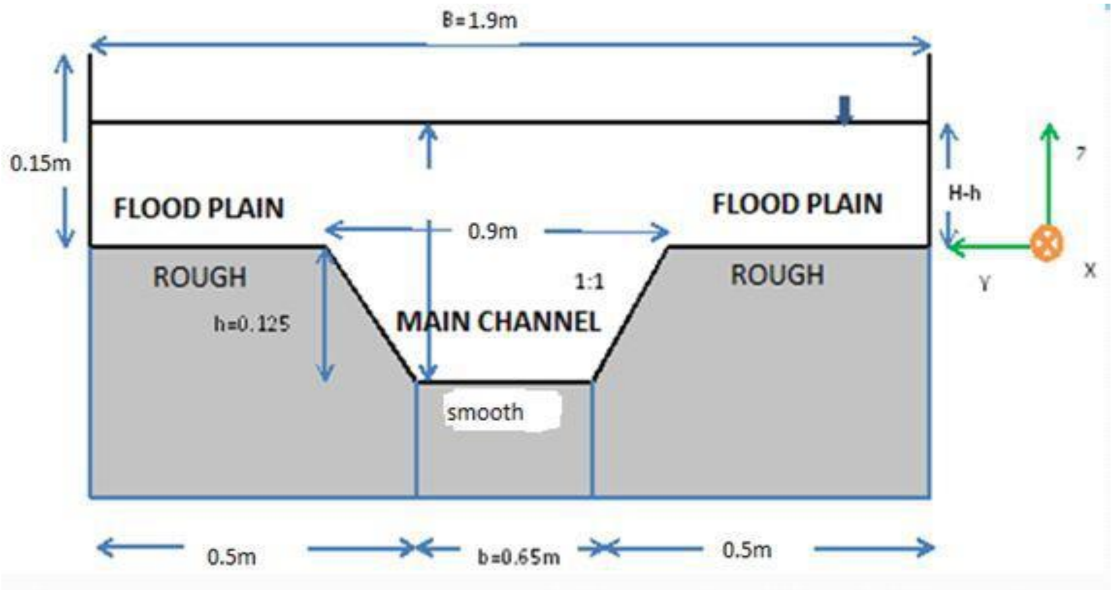


Fig 3.2 Cross Sectional View of Experimental Setup Of The Channel

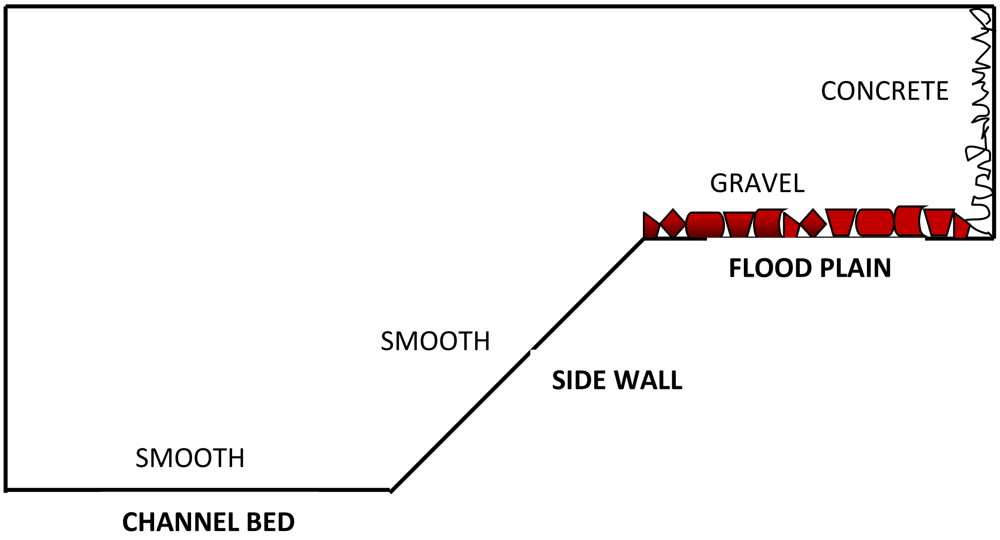


Fig 3.3 Half Cross Sectional View Of Trapezoidal Channel

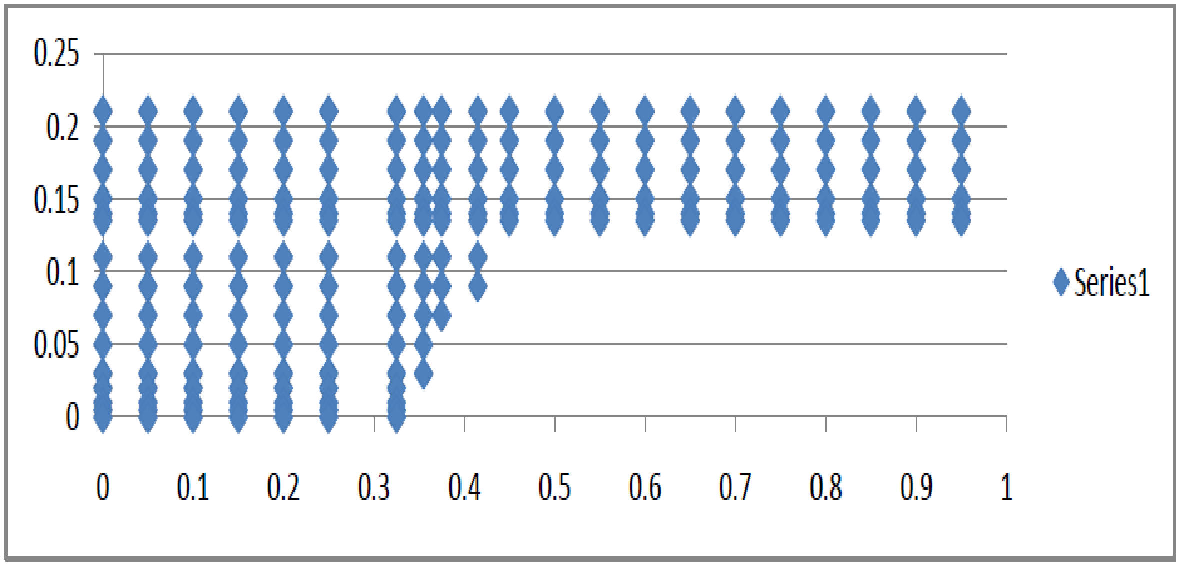


Fig 3.4 Typical Grid Points For Measurement Of Velocity

3.1 Description Of Numerical Model Parameters

In this study, Fluent, for model verification a Computational Fluid Dynamics simulation tool is used which is based on the three-dimensional form of Navies-Stokes equations. Generally a finite volume method (FVM) is used in CFD. Both structured and unstructured grids is used in Fluent .The governing equations are discretized in both space and time in free-surface modeling e.g. VOF (*15*) and height of liquid (HOL) or LES, which generally requires transient simulation. Hereford turbulence modeling Large Eddy Simulation model is used. The LES equations are discretized in both space and time. In this study the algorithms adopted to solve the coupling between pressure and velocity field is PISO, the pressure implicit splitting of operators use in Fluent (*16*). A no iterative solution method PISO is used to calculate the transient problem as it helps to converge the problems faster. When the residuals of the discretized transport equation reach a value of 0.001 or when the solution do not change with further iterations, the numerical solution is converged. To promote the convergence of the solution the changing variables are controlled during the calculations. For the simulations with an unsteady solver, the difference in the mass flow rates at the velocity inlet and pressure outlet is monitored to be less than 0.01% in the final solution. Furthermost, a number of extra time steps are added to verify the steadiness of the flow field in the final solution.

In this numerical simulation process there are four steps involved:

(a) Geometry setup of the experimental channel
(b) Creating the mesh for the geometry
(c) Set up physics
(d) Post-processing.

3.1.1 Geometry setup

The first step in CFD analysis is the creation of the geometry of the fluid flow region. A consistent frame of reference for coordinate axis was adopted for creation of geometry.

Here in coordinate system, X axis shows the lateral direction which indicates the width of channel bed, Y axis indicates the vertical component i.e., depth of water in the channel and Z axis indicates the direction of fluid flow. The water flowed along the negative direction of the z-axis.

During the model construction, the geometry is given names for different parts known as named selection. This is done to conduct analysis and for applying boundary condition upon a particular domain. Figure 4.3, 4.4, 4.5, 4.6 shows the geometrical entities used in the rigid vegetated channel.

The named selection includes these six parts:

1. Inlet
2. Outlet
3. Flood Plains
4. Channel Bed
5. Top symmetry

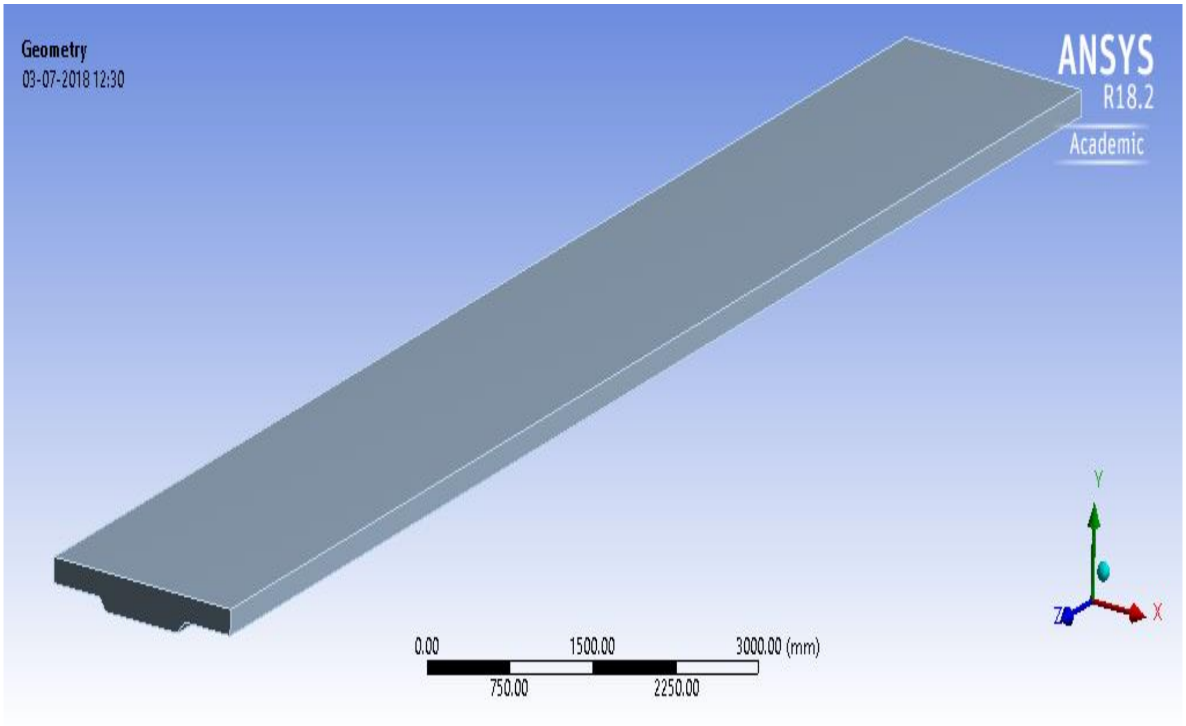


Fig 3.5 Geometry of the Channel

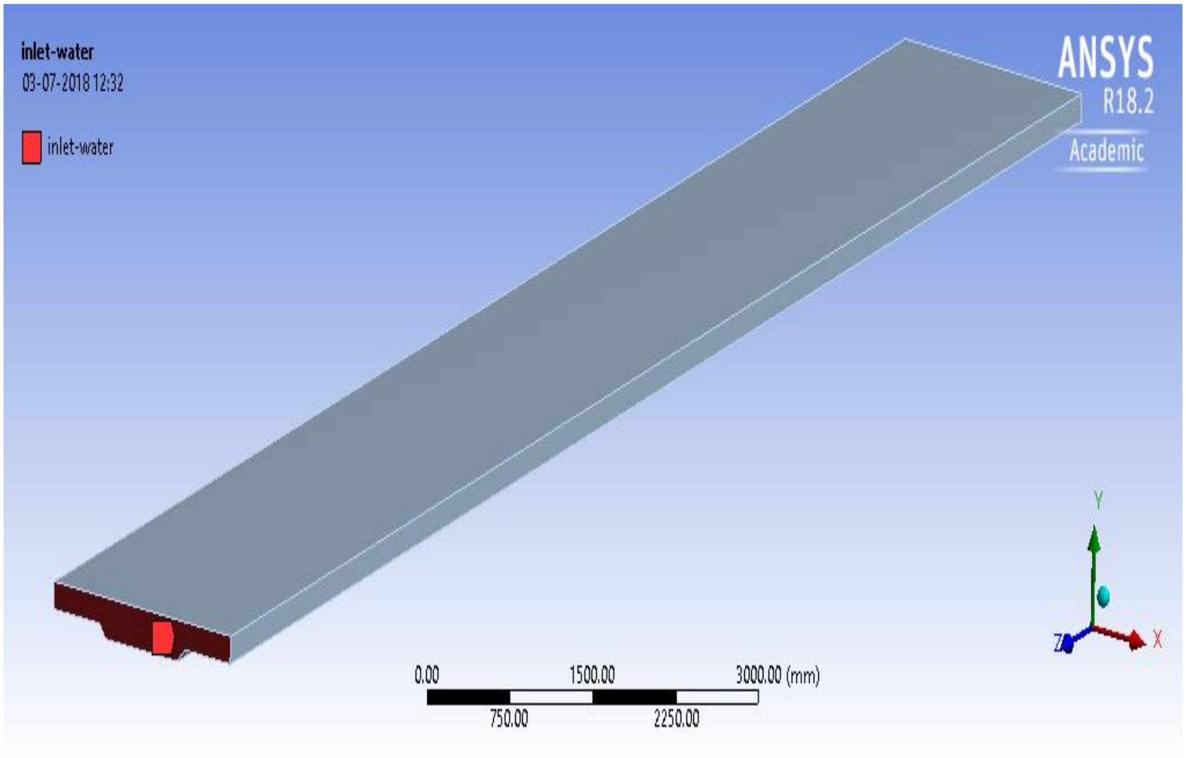


Fig 3.6 Water Inlet of the Channel

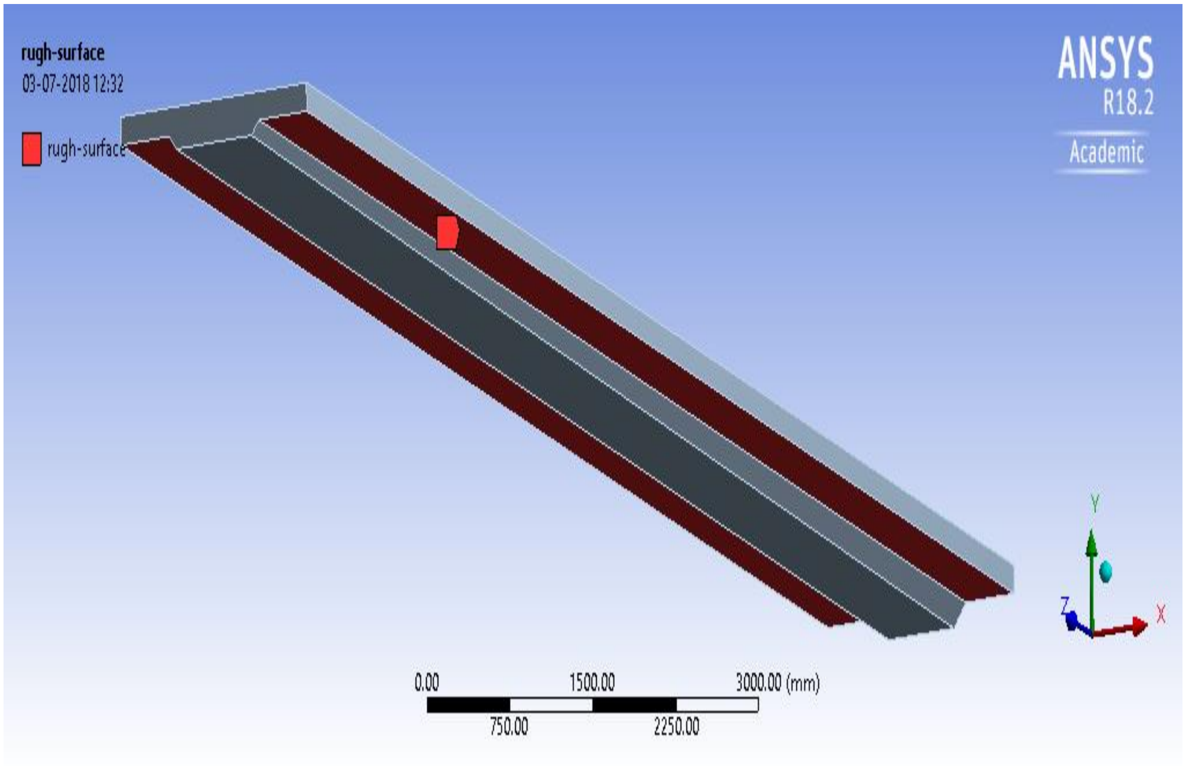


Fig 3.7 Flood Plains of the Channel

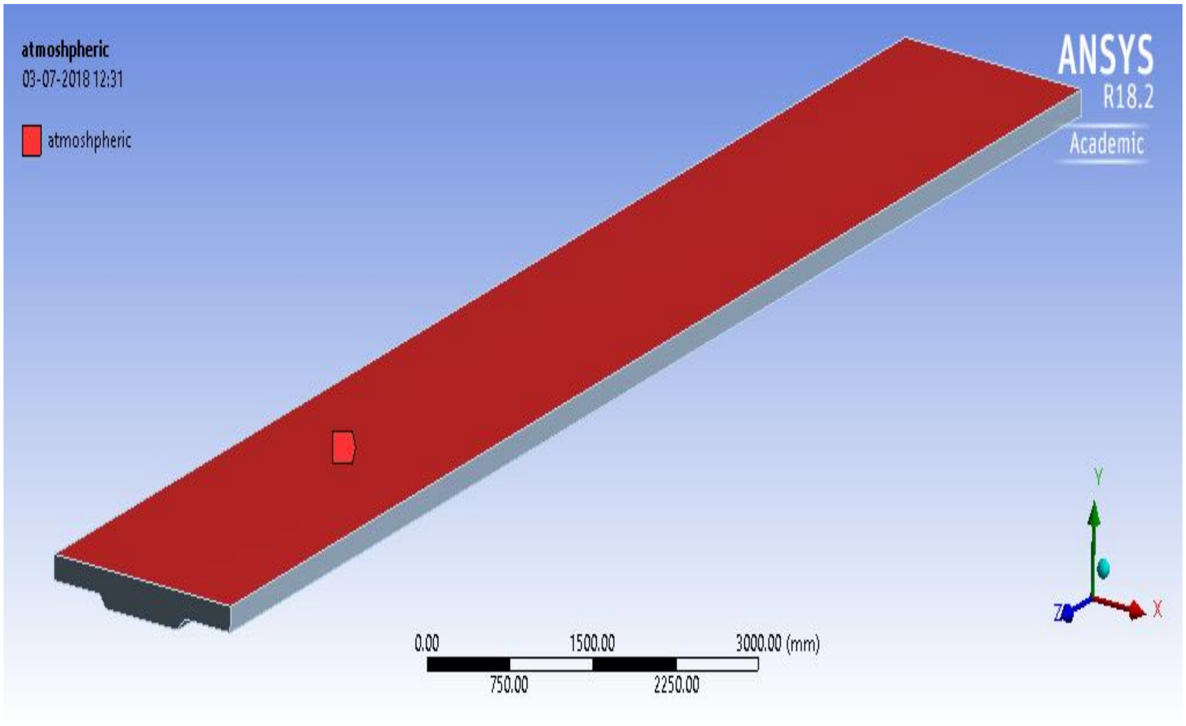


Fig 3.8 Water Surface of the Channel at Atmospheric Pressure

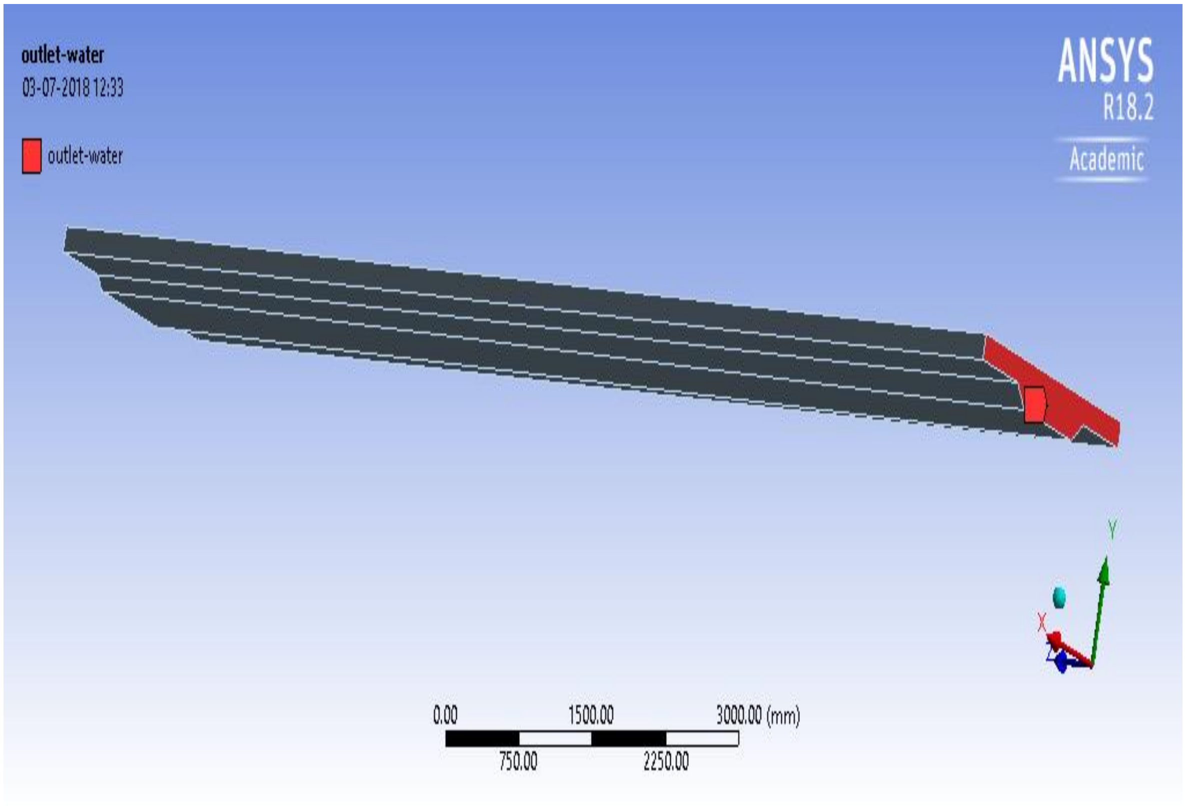


Fig 3.9 Water Outlet of the Channel

3.1.2 Mesh generation

Second and very most important step in numerical analysis is meshing of geometry. Meshing is described as discretizing or subdividing the geometry into the cells or elements at which the variables will be computed numerically. Meshing divides the continuum into finite number of nodes.

There are three different ways to descritize the fluid domain i.e. Finite element, Finite Volume and Finite Difference Method. Here finite volume method is used for descritization.

The Finite Volume method divides the domain into finite number of volumes. This method solves the discretization equation in the center of the cell and calculates some specified variables. The velocity value are calculated by taking it at the centre of each volume and adding all the volumes.

The next important thing in meshing is dense of meshing. It should not be too dense or too light meshing. Dense meshing consumes extra memory and takes alot of time. While light meshing gives results which are so much different from experimental results. So the meshing should be proper. Meshing plays very important role in giving meshing. So meshing should be dense near the walls where cylinders are present and not very dense in other parts.

Converging of a solution depends on meshing only. The meshing of the vegetated channel is shown in the Figure 3.10

.

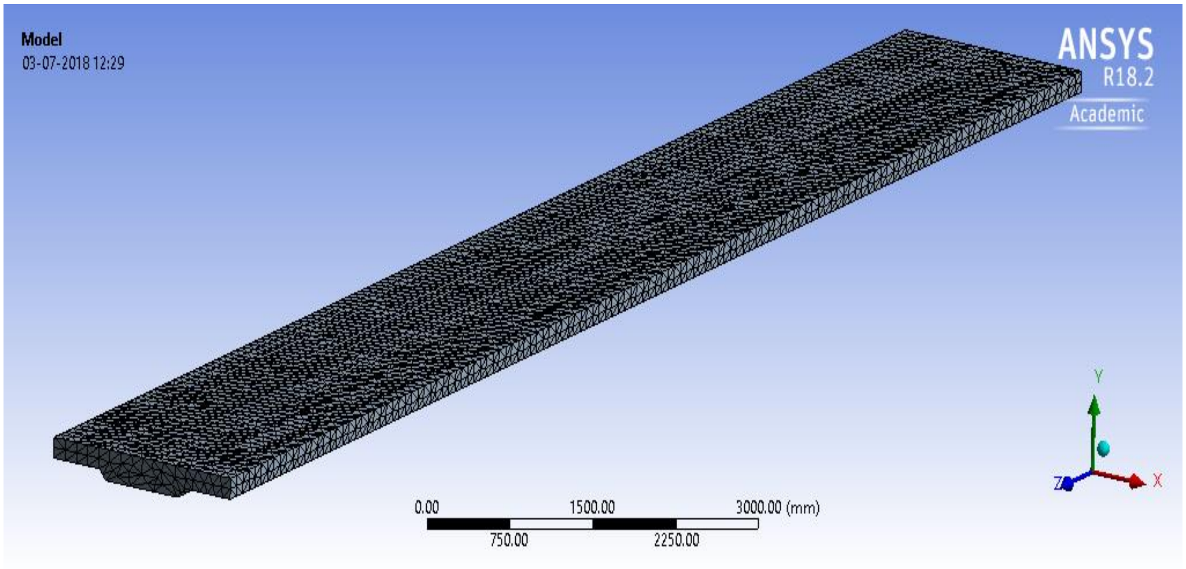


Fig 3.10 Meshing of the Open Channel

3.1.3 Setup physics

The next important thing in numerical simulation is setup physics. There are different things in this section.This consists of various models used for analysis, the initial and boundary conditions, the number of Eulerian phases, the properties of the materials. The model used in this is K-epsilon RNG(Re-Normalisation Group).

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.

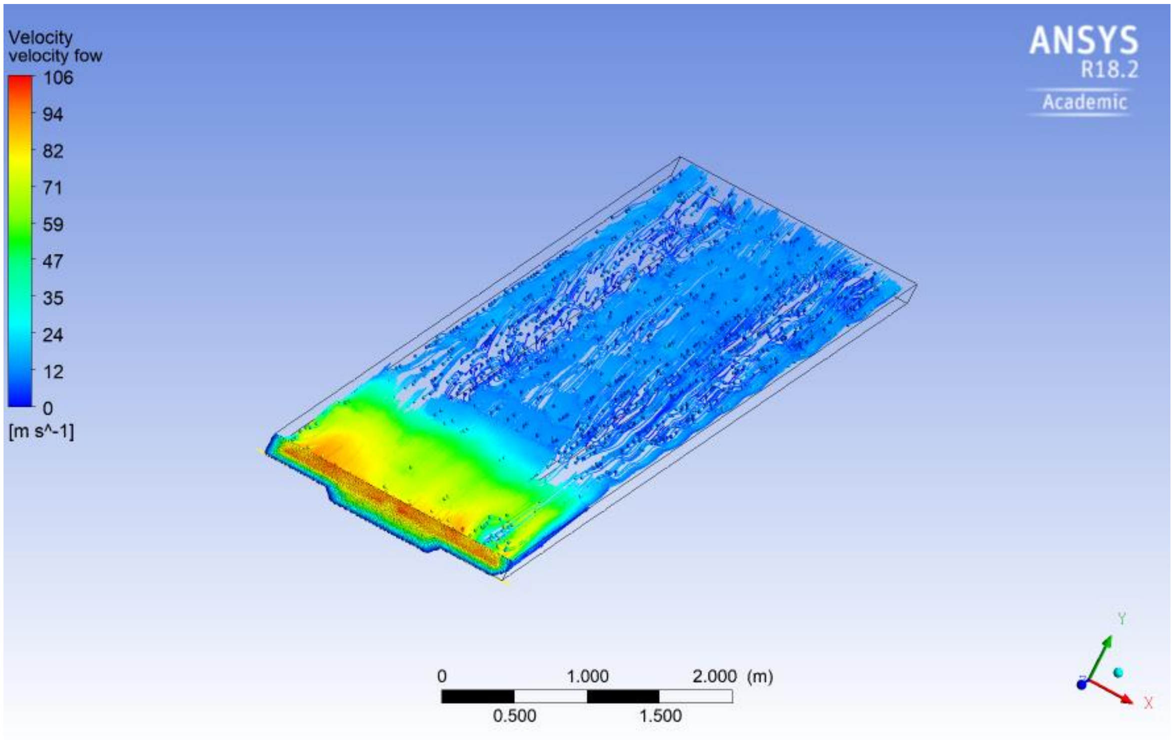


Fig 3.11 Velocity Contour of the whole Channel

4

RESULT AND DISCUSSIONS

Once normal depth conditions were established for a given discharge, point velocity measurements made across one section of the channel at z = 0.4h from the bed. At each lateral position, a number of readings were taken at constant intervals and then averaged to reduce error.

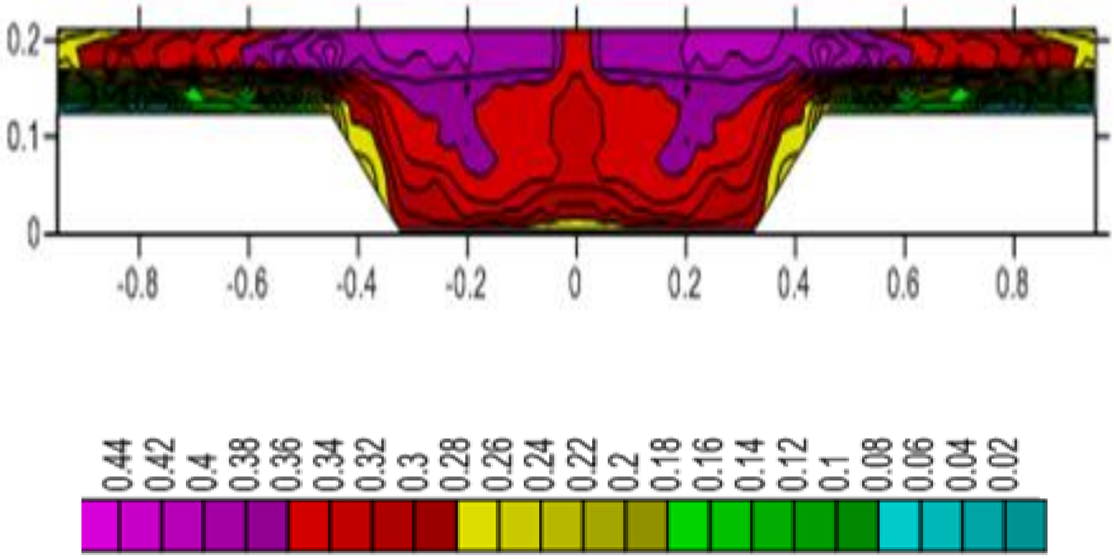


Fig 4.1 Velocity Contour Using Experimental Data

The resistance and velocity profiles of such channels are found to be changing with the flow depth. In a vegetative open channel flow, the average water velocity in the cross section tends to decrease at a higher rate, due to flow resistance from the stems and leaves of the vegetation which generally increases roughness of surfaces. Because of this complex nature, it is hard to develop a flow model based on theoretical calculations and derivations.

The velocity distribution at any point in any particular zone was assumed to follow logarithmic distribution involving the shear velocity at the foot of the normal drawn from the point to the corresponding wall. This method, in effect, amounts to considering that there is no momentum exchange across the planes passing through the bisectors of the base angles and parallel to the flow direction.

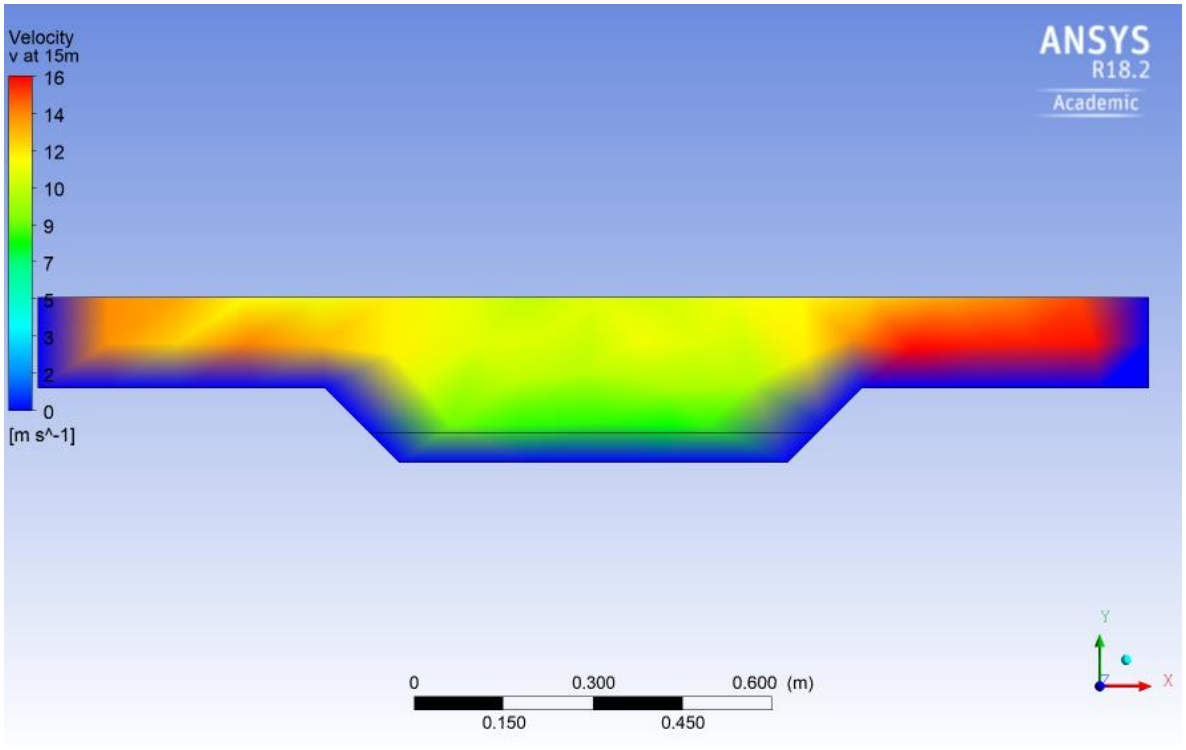


Fig 4.2 Velocity Contour Using Ansys

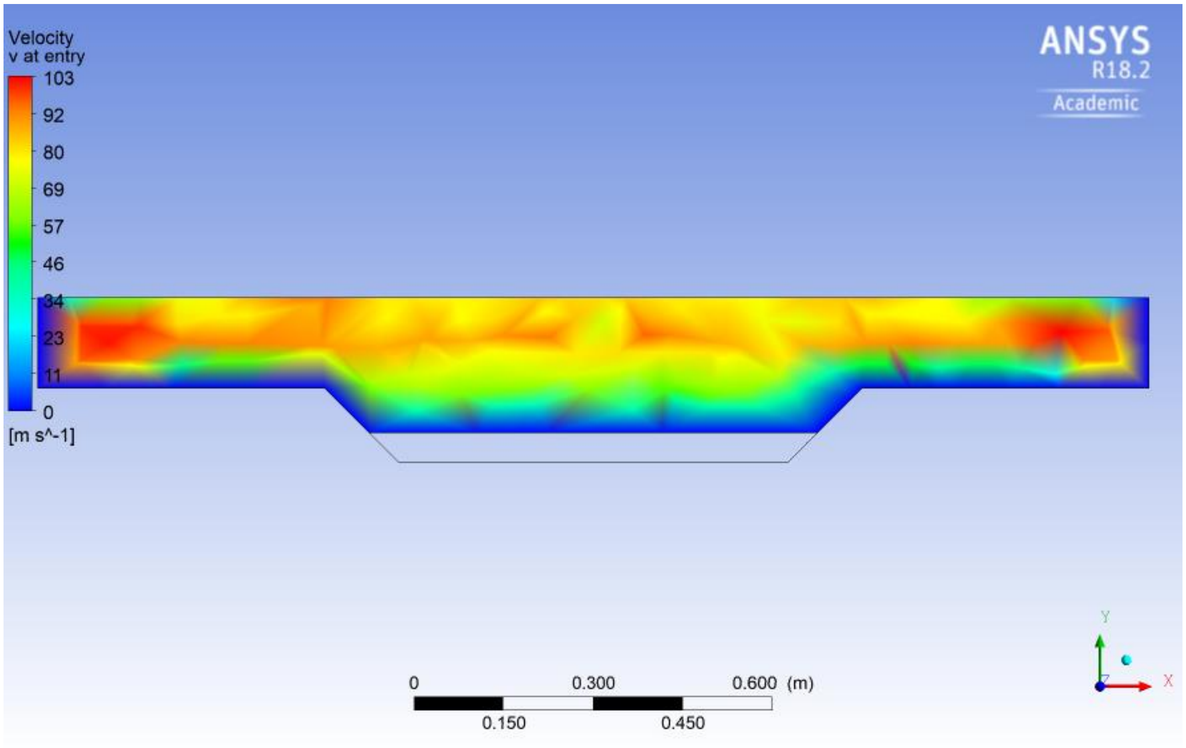


Fig 4.3 Velocity Contour At Entry

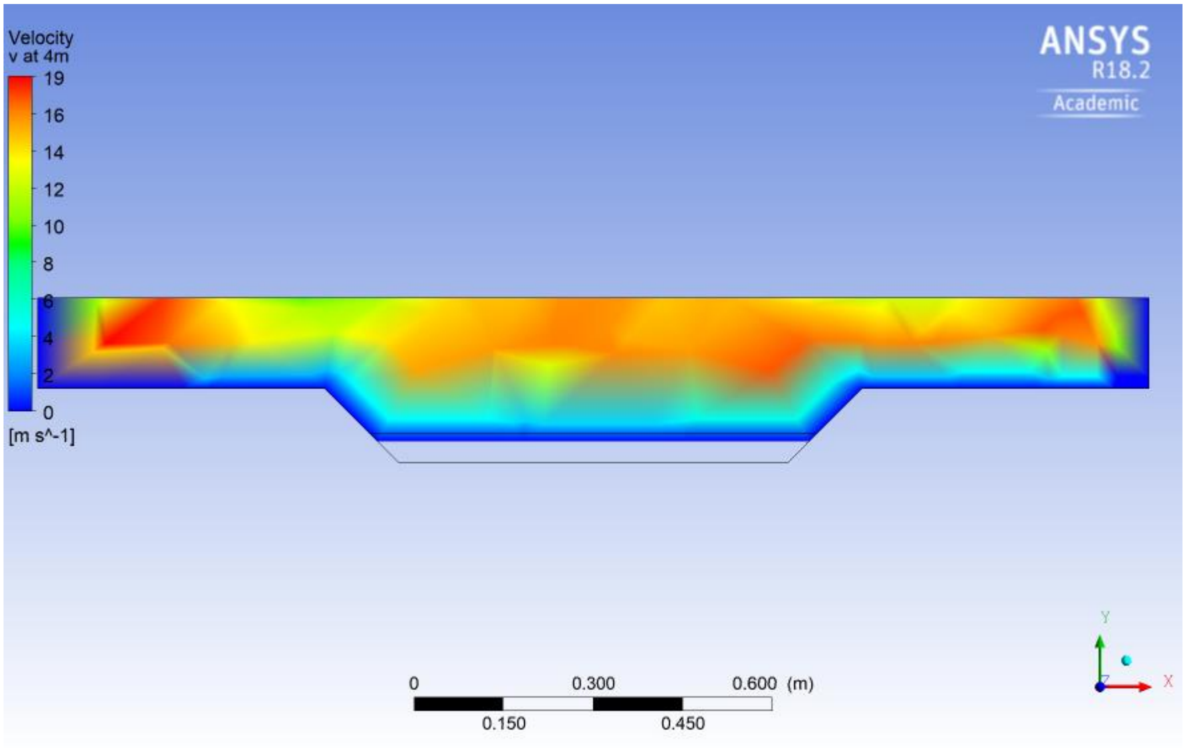


Fig 4.4 Velocity Contour at 4m

4.1 Estimation of Boundary Shear

Previous researchers have estimated boundary shear carried by various zones such as floodplain and main channel regions by conducting laboratory experiments on compound channels having different geometric and hydraulic parameters (e.g. α and β).It has been shown that the percentage shear force carried by different zones is a non- linear function of the percentage area occupied by that zone. In other words, % S_{fp} and % S_{mc} are usually power or exponential functions of % A_{fp} and % A_{mc} respectively. For compound sections having different width ratios, generally different relationships hold well and as in the present case the width ratio is in the range of 6.67-12 so a new expression has to be developed which would be applicable in the present case. In the next few paragraphs the sequential development of the expressions for different compound sections with different values of α is briefly outlined as a prelude to the latest developed model of boundary shear.

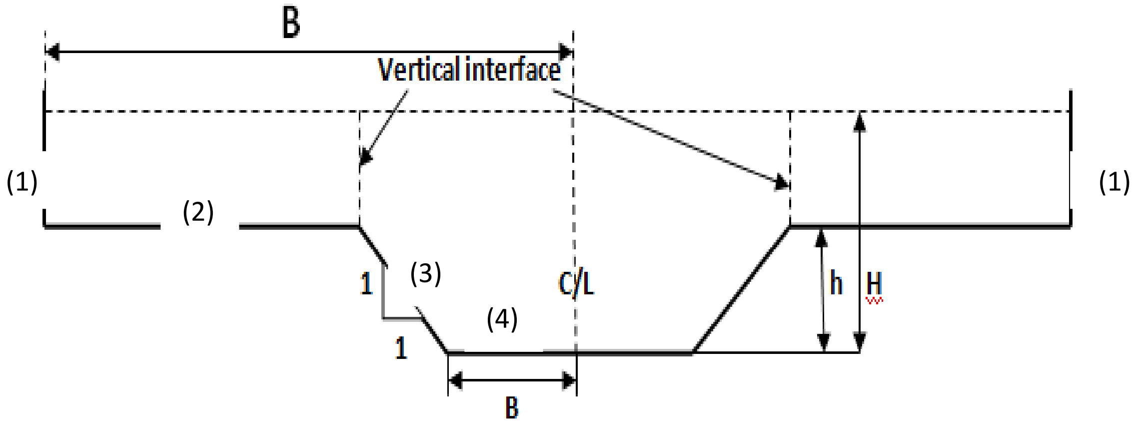


Fig 4.3 Detailed View Of Cross Section Of Channel

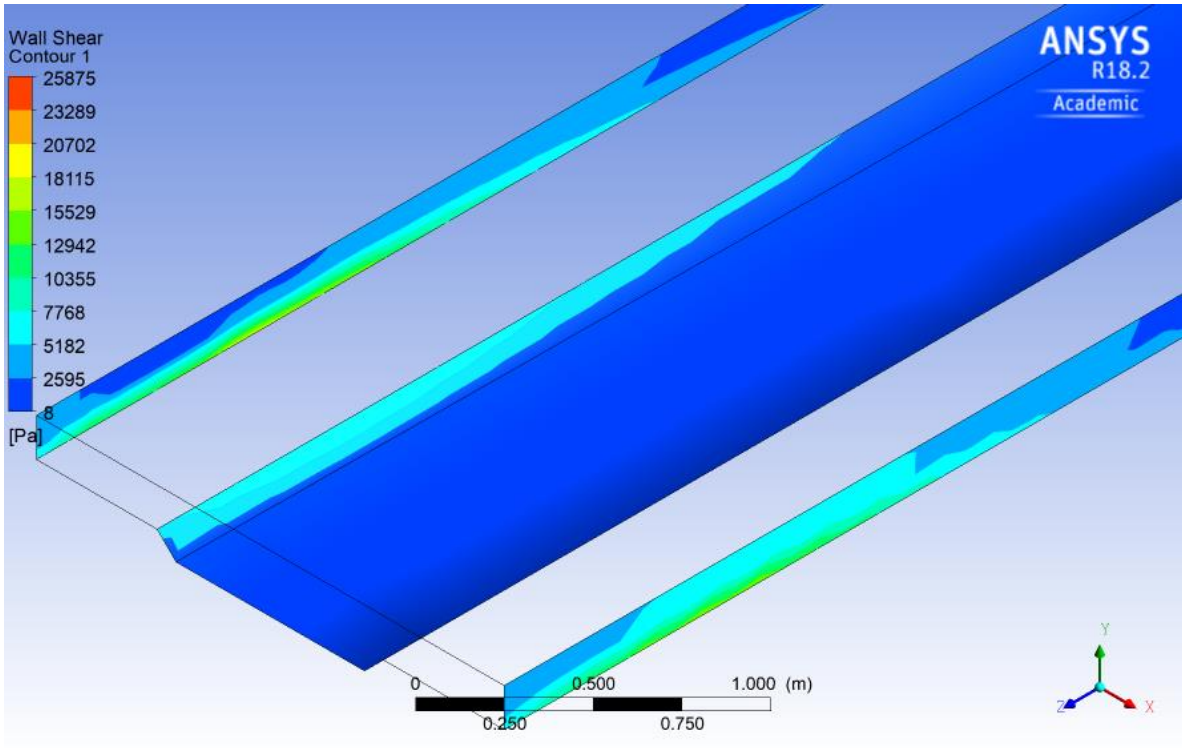


Fig 4.4 Wall Shear Contour

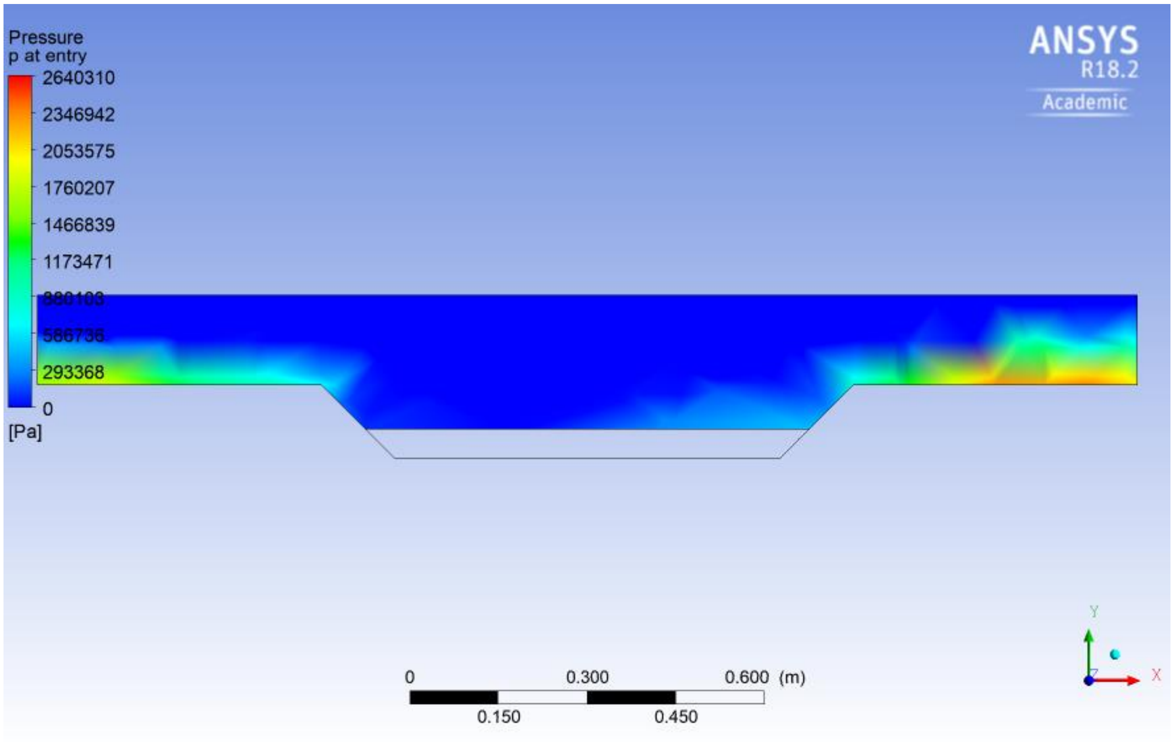


Fig 4.5 Pressure contour at entry

Various boundary elements comprising the wetted parameters are labeled as (1), (2), (3) and (4) in Fig.(4.3). Label (1) denotes the vertical wall(s) of floodplain of length ((H - h)) where H is the total depth of flow from main channel bed, and h is the depth of main channel. Label (2) denotes floodplain beds of length (B - b) , where B =1/ 2 (total width of compound channel), and b = 1/ 2 width or bed of main channel represented by label (4).

Label (3) denotes the inclined main channel walls of length (2h) . The shear stress distributions at each point of the wetted perimeter are numerically integrated over the respective sub-lengths of each boundary element (1), (2), (3), and (4) to obtain the respective boundary shear force per unit length for each element in the half section of the symmetric channel cross section. Twice the sum of the boundary shear forces for all the elements thus calculated in beds and walls of the compound channel gives the total shear force resisted in the whole compound section and is used as a divisor to calculate the shear force percentages carried by the boundary elements (1) through (4). The percentage of shear force carried by floodplains comprising elements (1) and (2) is represented as % S_{fp} and for the main channel comprising elements (3) and (4) is represented as % S_{mc} .
Following Knight and Demetriou, Knight and Hamed proposed an equation for % fp S for a compound channel section as :-

$$\%S_{fp} = 48(\alpha - 0.8)^{0.289}(2\beta)^{m} \qquad (1)$$

Equation (1) is applicable for the channels having equal surface roughness in the floodplain and main channel. For channels with non-homogeneous roughness,
the equation was improved by Knight and Hamed as:-

$$\%S_{fp}=48(\alpha-0.8)^{0.289}(2\beta)^{m}(1+1.02\sqrt{\beta}\lg\Upsilon) \qquad (2)$$

The exponent m can be evaluated from the relation

$$m=\frac{1}{0.75\ e^{0.38\alpha}} \qquad (3)$$

For homogeneous roughness section ($\Upsilon = 1$, where Υ is defined as ratio of floodplain roughness to the main channel roughness), Eq.(2) reduces to Eq.(1). The adequacy of the Eq.(2) for smaller width ratio channels has been shown in Ref.[2]. A regression analysis was also made by Khatua and Patra and they proposed an equation for $\% S_{fp}$ as

$$\%S_{fp}=1.23\beta^{0.1833}(38Ln\alpha+3.6262)(1+1.02\sqrt{\beta}\lg\Upsilon) \tag{4}$$

Knight and Hamed have shown that Eq.(2) is valid for compound channels having width ratio α up to 4, while Khatua and Patra have shown the validity of Eq.(4) for α up to 5.25. However when tested against very wide FCF channel ($\alpha = 6.67$) , significant errors in the estimation of boundary shear carried by flood plains resulted from both Eqs.(2) and (4). 90% and 70% errors were reported by Eqs.(2) and (4) respectively. So Khatua et al. based on regression analysis carried over three data series of Ref.[1] and five series of FCF-A channels and data of compound channel of Ref.[7] obtained a new relation for percentage shear carried by the flood plain as

$$\%S_{fp}=4.1045(\%A_{fp})^{0.6917} \tag{5a}$$

For non-homogeneously roughened channels as before the Eq.(5a) takes the form

$$\%S_{fp}=4.1045(\%A_{fp})^{0.6917}(1+1.02\sqrt{\beta}\lg\Upsilon) \tag{5b}$$

For rectangular channel and floodplains, Eq.(5a) can be expressed as

$$\%S_{fp}=4.105[100\beta(\alpha-1)/1+\beta(\alpha-1)]^{0.6917} \tag{5c}$$

where A_{fp} is the area of the floodplain. Equation (5a) results in minimizing the error in shear estimation for wide compound channels and is validated for the width ratio α up to 6.67. In view of the compound channel considered in the present case where the width ratio equals nearly 12, on the basis of the new experimental results obtained in higher width channels, the Eq.(5) is further modified as

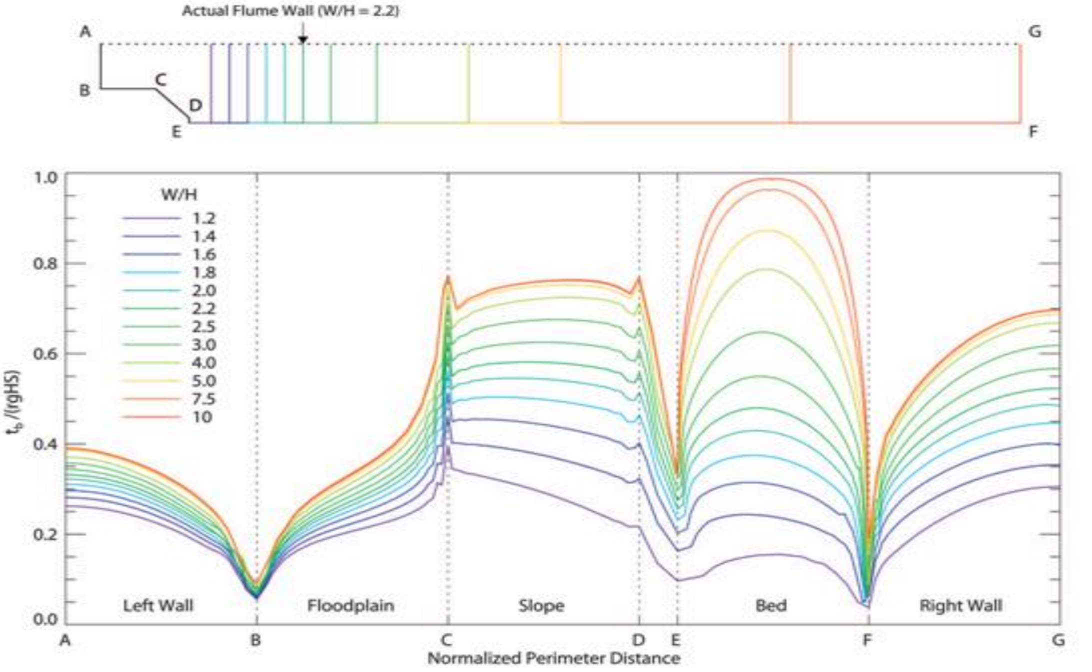
$$\%S_{fp}=3.3254(\%A_{fp})^{0-7467} \tag{6a}$$

Equation (6a) has been obtained by taking the present wide channel data and widest compound channel data from FCF series (i.e., for $\alpha = 6.67$). The regression analysis was done only by taking these two specific-standard data sets as these only correspond to compound channels with wide flood plains ($6.67 \leq \alpha \leq 11.96$) and R^2 value was obtained as 0.97. For non homogeneously roughened channel again the same factor suggested by Knight and Hamed is retained and for the present wide channel cases (6a) is written as

$$\%Sfp=3.3254(\%Afp)^{0\text{-}7467}(1+1.02\sqrt{\beta}\lg\Upsilon) \tag{6b}$$

For trapezoidal main channel and rectangular floodplains, in terms of non-dimensional parameters α , β , б and s , where “ s ” is the value of side slope of trapezoidal main channel (1: s ::V : H) the percentage of floodplain area can be simplified as

$$\%A_{fp}=\frac{\beta\alpha\delta-\beta(\delta+2s)}{\beta\alpha\delta+(1-\beta)(1+\delta)} \tag{7}$$



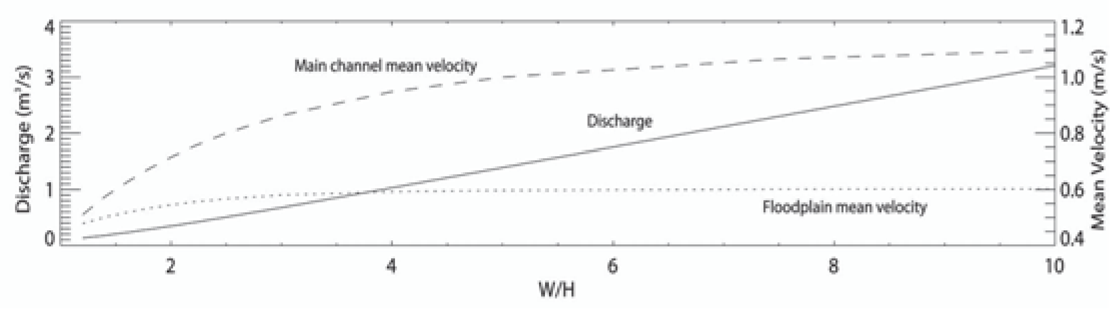


Fig.4.6(a) Color Calculated boundary shear stress and (b) mean flow velocity and discharge for flume channels with varying right wall locations. The channel geometry for each case is shown with no vertical exaggeration at the top of the figure. The actual flume geometry corresponds to the case *W*/*H*=2.2. The boundary shear stress is normalized by the depth-slope product, which is 1.96 N/m2.

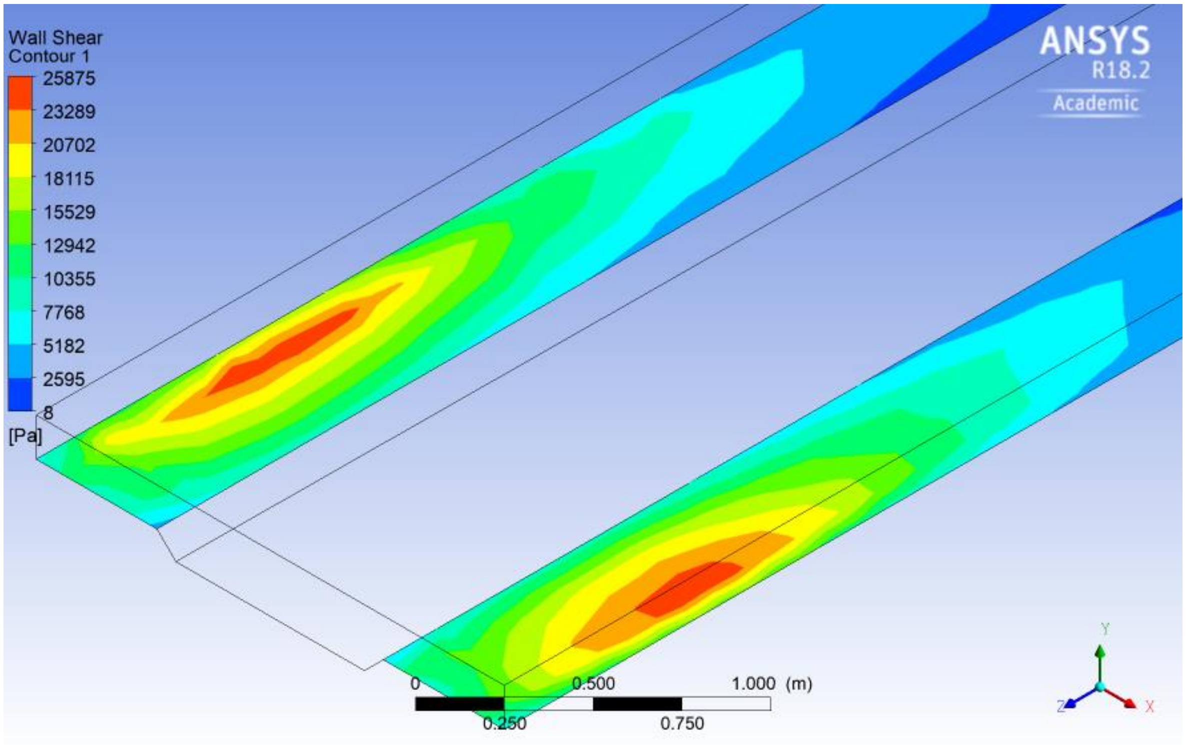


Fig 4.7 Shear at Flood Plains

The first scenario, shown in Fig. 4.7, depicts the variation in boundary shear stress profiles for 12 different positions of the right wall of the flume. In each of these cases, the depth, slope, roughness, and left bank geometry remains identical to the original flume geometry. The second scenario, shown in Fig.4. 8, presents a similar set of 10 cases in which the position of only the left wall of the flume is changed. These latter cases have floodplains of varying widths, which are roughened by cobbles having the same mean diameter and spacing as was present in the original flume.

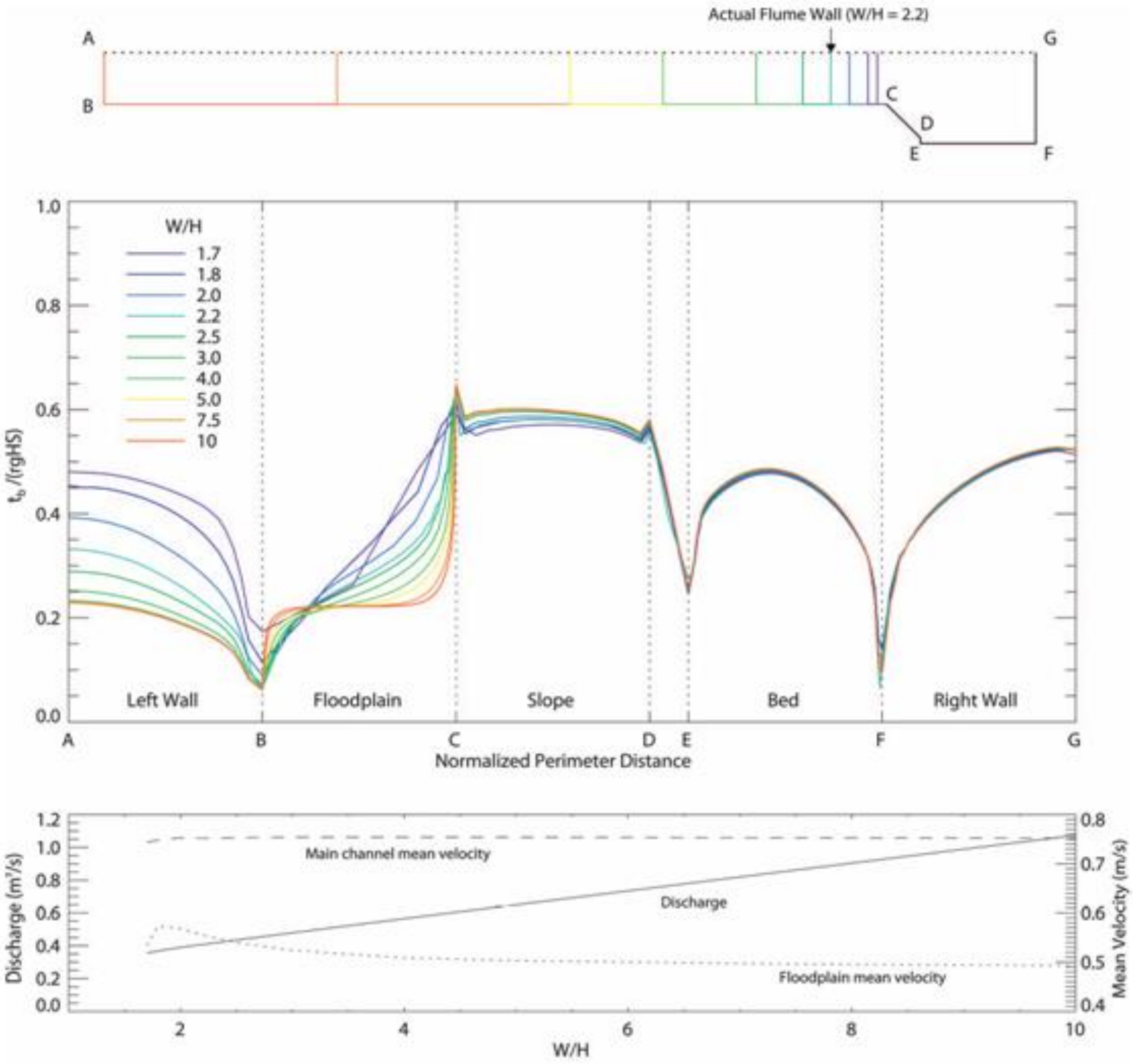


Fig. 4.8 (Color)(a) Calculated boundary shear stress; (b) mean flow velocity and discharge for flume channels with varying left wall locations. The boundary shear stress is normalized in the same manner as was done in Fig. (3.11). For reference, the floodplain depth-slope product is 57% of $\rho g \sin(\theta) H$. The asymptotic limit of stress on the floodplain is 39% of the floodplain depth-slope product due to the drag on the floodplain cobbles. Note that varying the width of the floodplain has essentially no effect on the stress on the bed and the right wall of the channel.

The calculated discharge for all of the cases varies as function of width and is shown in Figs. 3.10b and 3.11b. Figures 3.10b and 3.11b also contain the mean velocity and area for the floodplain and main channel. The floodplain and main channel portions of the channel are defined to be the respective sections to the left and right of corner "C" in Figs. 3.10 and 3.11. Calculated Reynolds numbers indicate the flow is turbulent for all cases. To facilitate comparisons between the boundary shear stress profiles of the cases in Figs. 3.10a and 3.11a, the perimeter distance *x*-axis has been normalized by the length of each segment of the boundary. Note in Fig. 3.10 that the walls, floodplain, and slope have the same dimensions; only the bed segment of the wetted perimeter changes length with each case. Similarly, in Fig. 3.11, the floodplain segment is the only portion of the wetted perimeter that changes dimension with each case. Also, note in Fig. 3.10 that as the flume widens and the hydraulic radius increases, there is a corresponding increase in the average boundary shear stress due to the integral constraint that the average boundary shear stress plus the drag stress on the cobbles equals $\rho g \sin(\theta) R$. Similar changes in the average stress are not immediately apparent in Fig. 3.11, because the geometric variations of that scenario result in only minor changes in the hydraulic radius with each case.

There are several common characteristics to the calculations presented in Figs. 3.10 and 3.11. In both geometric scenarios, the boundary shear stress in the central portion of the width-varying segment bed segment in Fig. 3.10, floodplain segment in Fig. 8 does not reach its asymptotic limit until the width-to-local-depth ratio is 10. For the scenario shown in Fig. 3.11, this limit corresponds to the depth-slope product, $\rho g \sin(\theta) H$. For the changing floodplain scenario Fig. 3.11, the asymptotic limit of boundary shear stress on the cobble roughened floodplain is the flood plain depth-slope product minus the drag stress on the cobbles. In the widest cases in Fig. 3.11, the drag stress is 39% of the floodplain depth-slope product.

A second feature common to both geometric scenarios shown in Figs. 3.10 and 3.11 is that the stress in the three sub-180° corners B, E, and F is very insensitive to the width to depth ratio. This is because corner stresses are primarily controlled by the geometry specifically, the radius of curvature of the corner. Rounder flume corners would have produced more gradual transitions in the stress between the linear segments of the flume boundary. Another interesting feature shared by the two sets of calculations, is that the stress on the opposite side of the channel from the moveable wall is relatively insensitive to the width-to-depth ratio. This feature is strongly present in the set of calculations shown in Fig. 3.11a and weakly present in the scenario shown in Fig. 3.10a.

The main reason the stress on the left side of the channels in Fig. 3.11a is less sensitive to changes in the width to depth ratio than the stress on the right side of the channels is because the high roughness of the cobble-roughened floodplain dominates the near-bank flow and boundary shear stress fields. In Fig. 3.11a, the boundary shear stress on the bed and right walls is insensitive to changes in floodplain width, because the flow in the main channel is controlled primarily by the flow depth and geometry of the right side of the channel. Figs. 3,10b and 3.11b show similar trends for the mean velocity of the two “halves” of the channel. The mean velocity in the “half” of the channel that does not vary in width quickly reaches a constant value.

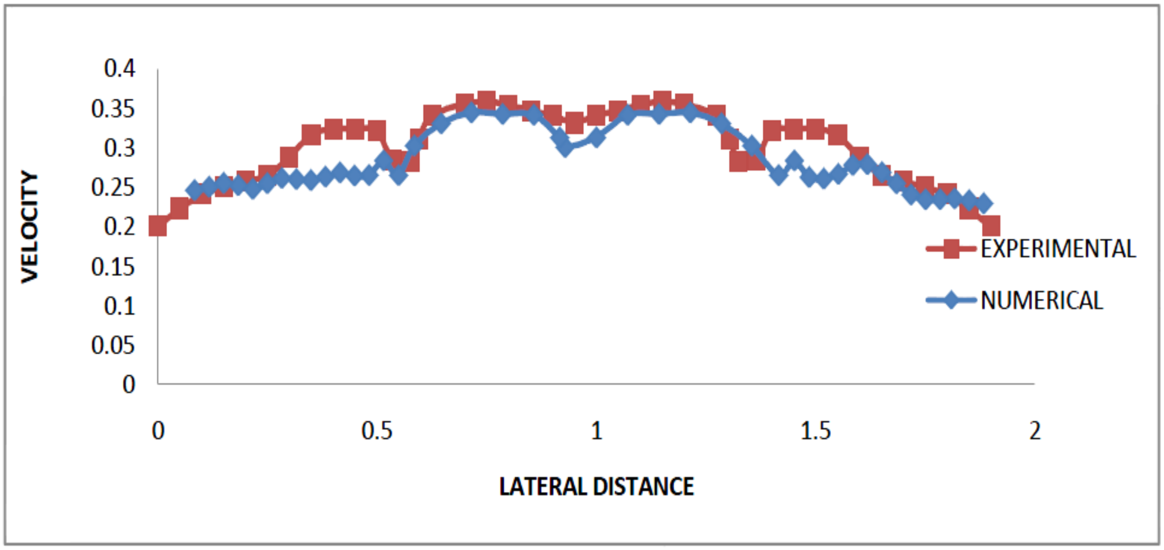


Fig 4.9 Validation of Experimental and Numerical Data

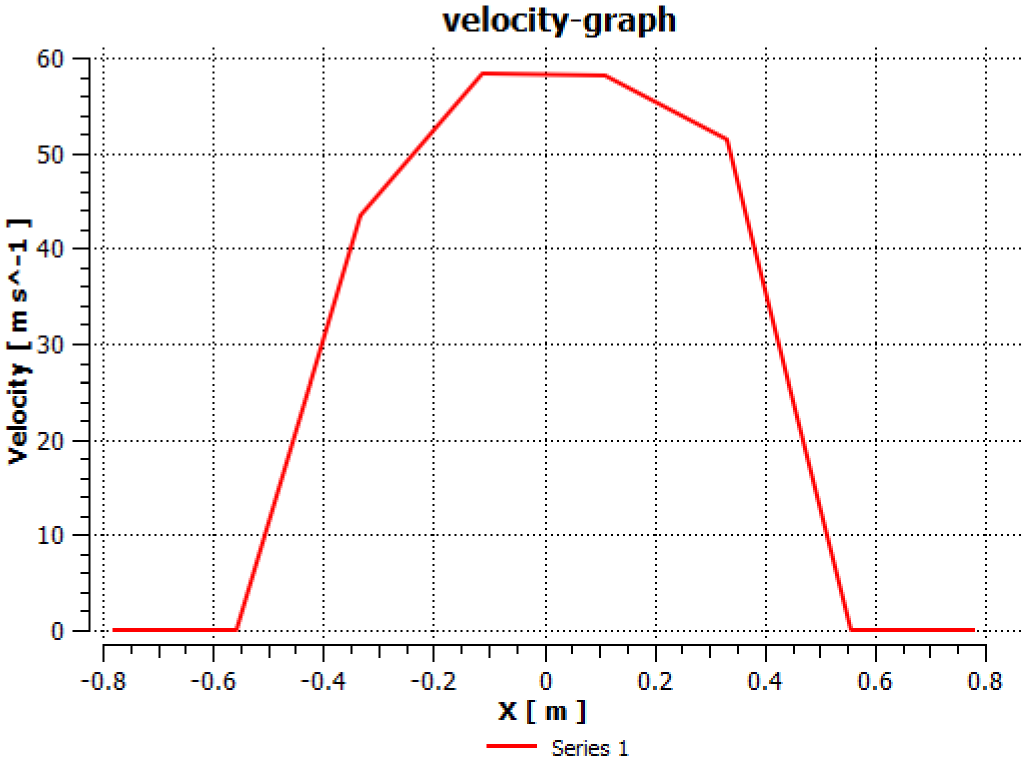


Fig 4.10 Distance vs Velocity graph

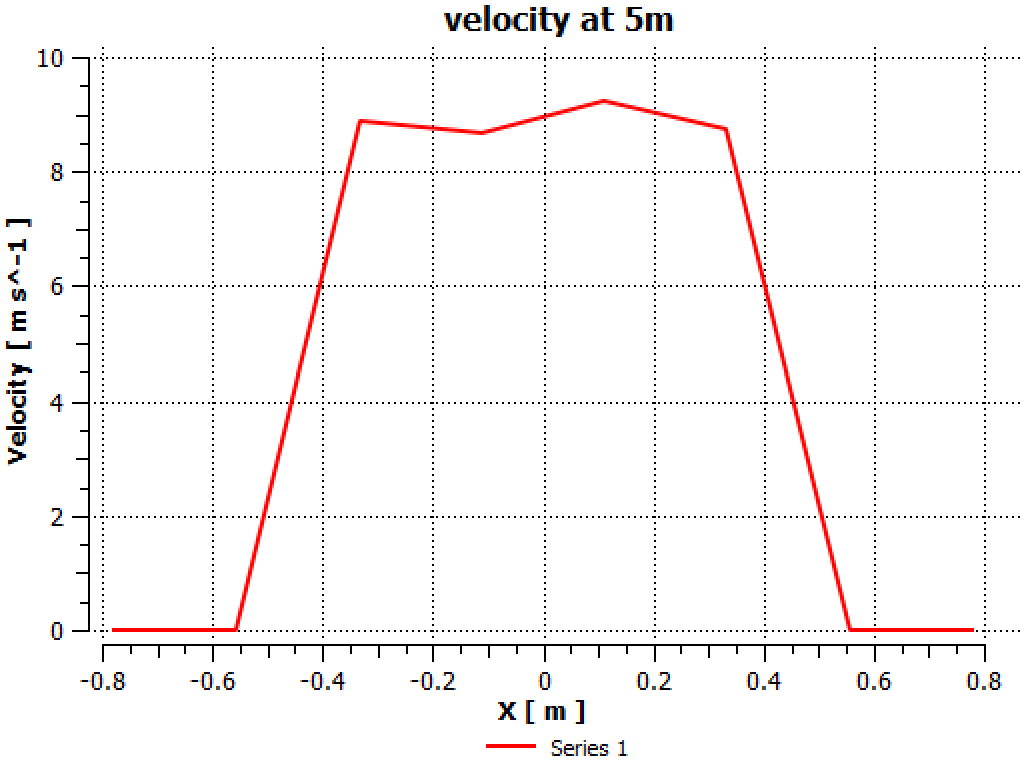


Fig 4.11 Distance Vs Velocity graph at 5m

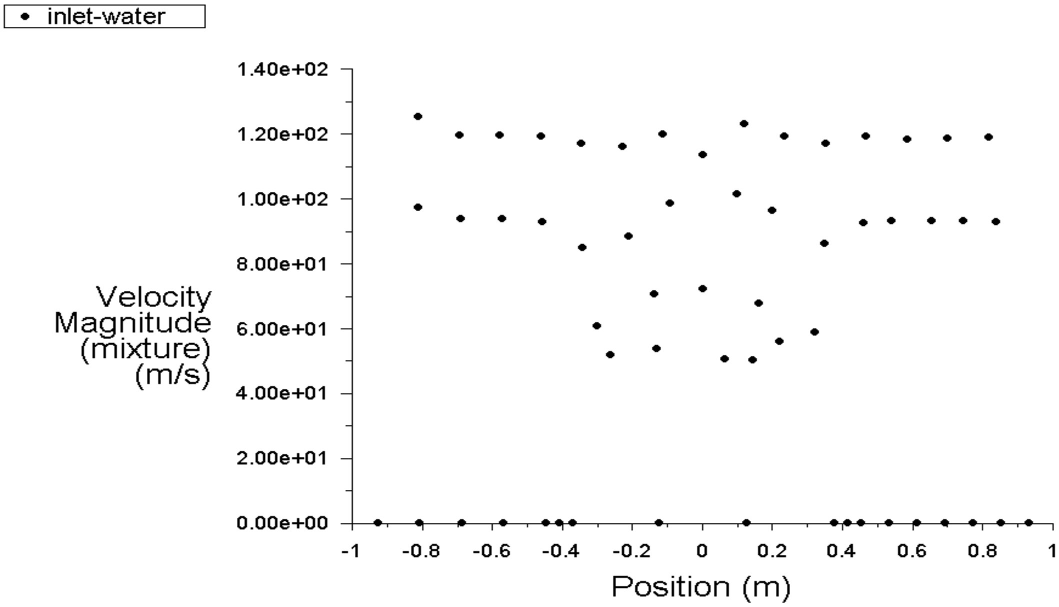


Fig 4.12 Velocity Magnitude Plot

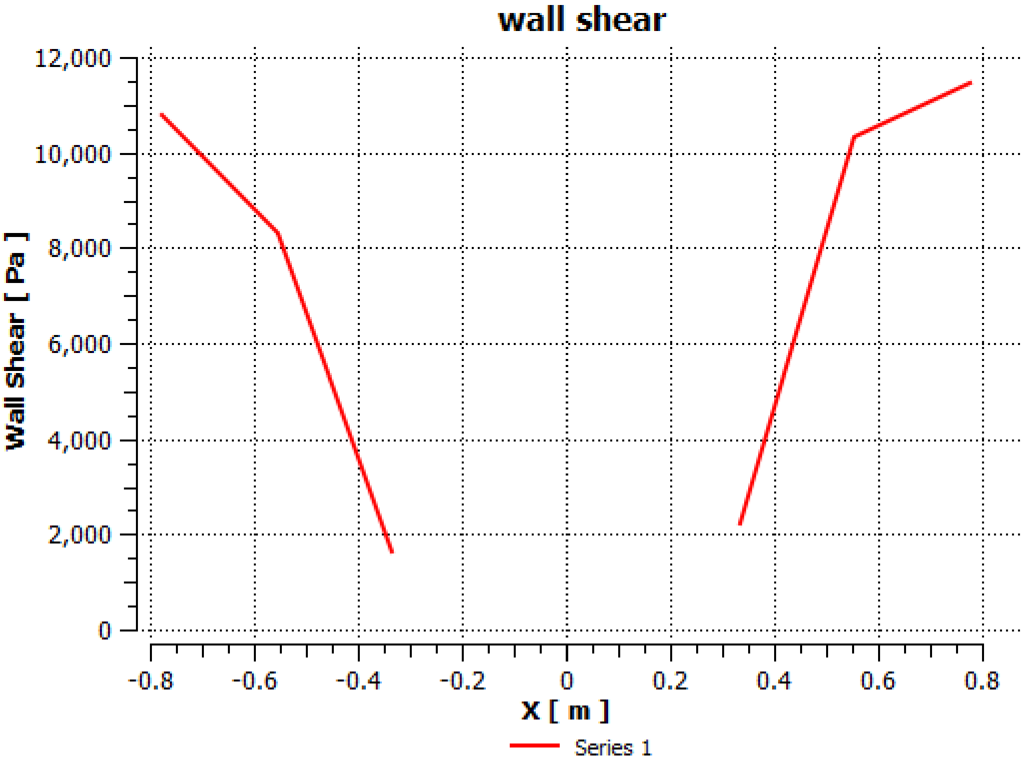


Fig 4.13 Boundary Shear at entry

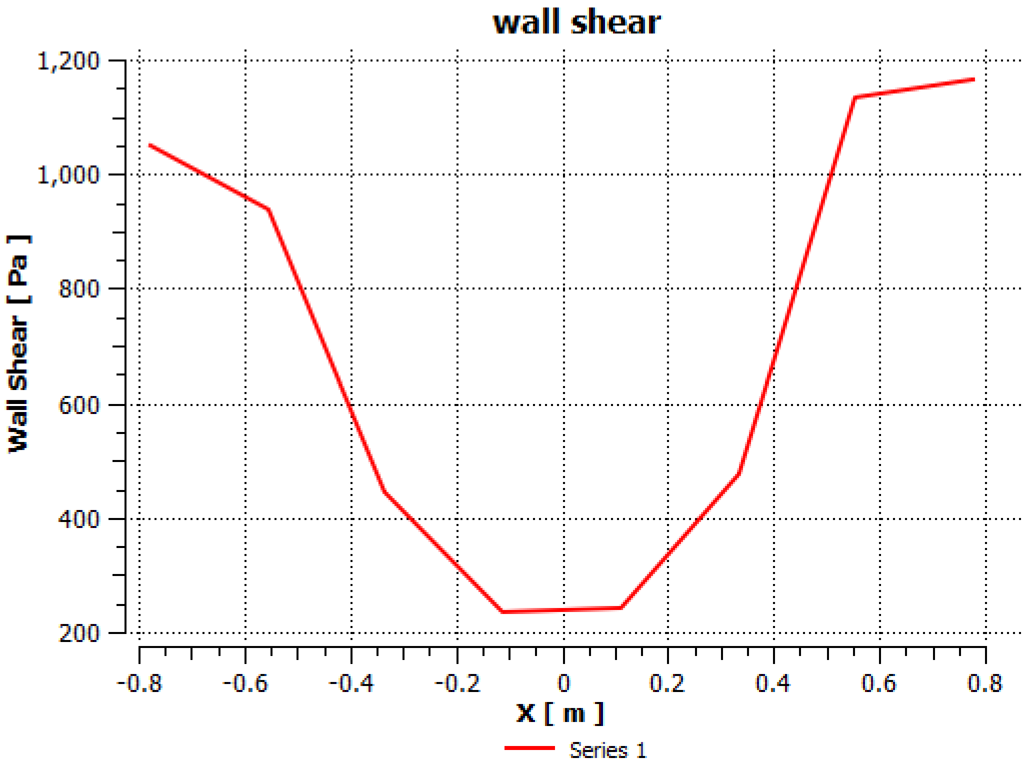


Fig 4.14 Boundary Shear at 5m

5

CONCLUSION

- The overall depth-averaged velocity in main channel increases with the increase in differential roughness (i.e. the percentage of flow increases with the increase in differential roughness), whereas decreases in floodplain region.

- The variation in depth-averaged velocity, in main channel and flood plain region is minimum in case of differential roughness (ɤ)=1. The variation increases with the increase in differential roughness.

- The overall discharge found to increase with the increase in depth of flow and decrease with the increase in differential roughness, which may be attributed to the fact that at higher depth of flow, the effect of differential roughness as well as that of the momentum transfer between main channel and flood plain, decreases.

- The concentration of maximum velocity contour is always found in main channel. The concentration found to decrease with the increase in depth of flow.

- The velocity variation on floodplain is found to be maximum for lowest depth of flow, then gradually stabilizing with the increase in depth of flow, whereas the variation is reverse in main channel.

- The overall range of variation in velocity is found to increase with the increase in differential roughness.

- The percentage of flow in main channel is found to increase with the increase in differential roughness, which may be attributed to that fact that the resistance to flow, offered by floodplain in comparison to main channel increases with the increase in differential roughness value (as main channel is smoother than floodplain).

- Fresh experiments conducted in a straight compound channel with very large width ratio ($\alpha = 11.96$) reveal that the shear force carried by the floodplains has a nonlinear relation with area covered by the floodplains and a new mathematical expression is derived on the basis of regression analysis with the coefficient of determination R^2 value of 0.967.

- The boundary shear stress distribution and division curves of trapezoidal channels with the best hydraulic section were obtained using mean bed and sidewall shear stresses. The determined division curves are significantly deformed due to the generation of secondary currents. For trapezoidal channels with the best hydraulic section, the mean bed and sidewall shear stresses are identical. Also, for these kinds of channels, maximum bed and sidewall shear stresses are approximately equivalent to two times the mean bed and sidewall shear stresses. The analytical results agreed well with the experimental measurements and the results of former investigations.

Future Outlook

The present work leaves a wide scope for future investigators to explore many other aspects of a vegetated open channel analysis. The future scope of the present work may be summarized as:

1. Trapezoidal channel with vegetation is used here. So converging channel bed can be used for further studies.

2. K-epsilon model is used in the FLUENT to carry the work. LES, k-ω, RSM models can be used to simulate various channel geometry with different hydraulic conditions.

3. K-epsilon model can be used for other hydraulic and geometrical conditions.

4. The results obtained here with trapezoidal channel can be compared with converging, diverging channel and can be applied to the natural channels.

REFERENCES

[1] Knight D. W., Demetriou J. D. Flood plain and main channel flow interaction[J]. **Journal of Hydraulic Engineering, ASCE,** 1983, 109(8): 1073-1092.

[2] Knight D. W., Hamed M. E. Boundary shear in symmetrical compound channels[J]. **Journal of Hydraulic Engineering, ASCE,** 1984, 110(10): 1412-1430.

[3] Khatua K. K., Patra K. C. Boundary shear stress distribution in compound open channel flow[J]. **ISH Journal of Hydraulic Engineering,** 2007, 13(3): 39-54
.

[4] Khatua K. K., Patra K. C. And Mohanty P. K. Stage-discharge prediction for straight and smooth compound channels with wide floodplains[J]. **Journal of Hydraulic Engineering, ASCE,** 2012, 138(1): 93-99.

[5] Khatua K. K. Interaction of flow and estimation of discharge in two stage meandering compound channels[D]. Doctoral Thesis, Rourkela, India: National Institute of Technology, 2008.

[6] Myers W. R. C. Velocity and discharge in compound channels[J]. **Journal of Hydraulic Engineering, ASCE,** 1987, 113(6): 753-766.

[7] Bhowmik N. G., Demissie M. Carrying capacity of flood plains[J]. **Journal of the Hydraulics Dvision, ASCE,** 1982, 108(HY3): 443-453.

[8] Cheng, N. S., and Nguyen, H. T. (2011). “Hydraulic radius for evaluating resistance induced by simulated emergent vegetation in open-channel flows.” **J. Hydraul. Eng.**,*137(9), 995–1004.*

[9] Carollo, F. G., Ferro, V., and Termini, D. (2002). “Flow velocity measurement in vegetated channels.” **J. Hydraul. Eng. 128~7!, 664–673.**

[10] Darby, S. E. (1999). “Modelling effect of riparian vegetation on flow resistance and flood potential.” **J. Hydraul. Eng. 125~5!, 443–454.**

[11] Fenzl, R. N., and Davis, J. R. (1964). “Hydraulic resistance relationships for surface flows in vegetated channels.” **Trans. ASAE, 7~1!, 46–55.**

[12] Ferreira, R. M. L., Ricardo, A. M., and Franca, M. J. (2009). “Discussion of Laboratory investigation of mean drag in a random array of rigid, emergent cylinders by Yukie Tanino and Heidi M. Nepf.” **J. Hydraul. Eng.,135(8), 690–693.**

[13] Fischer-Antze.T, Stoesser.T , and Olsen.N. R. B.(2001), “3D numerical modelling of open channel flow with submerged vegetation,” **Journal of Hydraulic Research, vol. 39, no. 3, pp. 303–310, 2001.**

[14] Ishikawa, Y., Mizuhara, K., and Ashida, S. (2000). “Effect of density of trees on drag exerted on trees in river channels.” **Eurasian J. For. Res., 5(4), 271–279.**

[15] James, C. S., Birkhead, A. L., Jordanova, A. A., and O“Sullivan, J. J. (2004). “Flow resistance of emergent vegetation.” **J. Hydraul. Res., 42(4), 390–398.**

[16] James, C. S., Goldbeck, U. K., Patini, A., and Jordanova, A. A. (2008).“Influence of foliage on flow resistance of emergent vegetation.” **J. Hydraul. Res., 46(4), 536–542.**

[17] Kothyari, U. C., Hayashi, K., and Hashimoto, H. (2009). “Drag coefficient of unsubmerged rigid vegetation stems in open channel flows.” **J. Hydraul. Res., 47(6), 691–699.**

[18] Kouwen, N., Unny, T. E., and Hill, H. M. (1969). “Flow retardance in vegetated channels.” **J. Irrig. Drain. Eng., 95~2!, 329–344.**

[19] Kowobary, T. S., Rice, C. E., and Garton, J. E. (1972). “Effect of roughness elements on hydraulic resistance for overland flow.” **Trans. ASAE, 15~5!, 979–984.**

[20] Lee, J. K., Roig, L. C., Jenter, H. L., and Visser, H. M. (2004). “Drag coefficients for modeling flow through emergent vegetation in the Florida Everglades.” **Ecol. Eng., 22(4–5), 237–248.**

[21] Liu, D., Diplas, P., Fairbanks, J. D., and Hodges, C. C. (2008). “An experimental study of flow through rigid vegetation.” **J. Geophys. Res.,[Earth Surf.], 113 F04015.**

[22] M. Ghisalberti and H. M. Nepf, “Mixing layers and coherent structures in vegetated aquatic flows,” **Journal of Geophysical Research: Oceans, vol. 107, no. 2, pp. 3-1–3-11, 2002.**

[23] Petryk, S., and Bosmajian, G. (1975). “Analysis of flow through vegetation.” **J. Hydraul. Div., 101,HY7!, 871–884.**

[24] Stoesser, T., Kim, S. J., and Diplas, P. (2010). “Turbulent flow through idealized emergent vegetation.” **J. Hydraul. Eng., 136(12), 1003–1017.**

[25] Stone, B. M., and Shen, H. T. (2002). “Hydraulic resistance of flow in channels with cylindrical roughness.” **J. Hydraul. Eng. 128~5!, 500– 506.**

[26] Tanino, Y., and Nepf, H. M. (2008). “Laboratory investigation of mean drag in a random array of rigid, emergent cylinders.” **J. Hydraul. Eng., 134(1), 34–41.**

[27] Thompson, G. T., and Roberson, J. A. (1976). “A theory of flow resistance for vegetated channels.” **Trans. ASAE, 19~2!, 288–293.**

[28] Tsihrintzis, V. A. (2001). “Discussion of Variation of roughness coefficients for unsubmerged and submerged vegetation”. **J. Hydraul. Eng., 127(3), 241–244.**

[29] Tsujimoto, T., and Kitamura, T. (1990). “Velocity profile of flow in vegetated-bed channels.” **KHL Progressive Rep. No. 1, Kanazawa University, Japan.**

[30] Rowinski PM, Kubrak J. (2002) “A mixing-length model for predicting vertical velocity distribution in flows through emergent vegetation.” **Hydrol Sci J 2002;47(6): 893–904.**

[31] Wu, F. C., Shen, H. W., and Chou, Y. J. (1999). “Variation of roughness coefficients for unsubmerged and submerged vegetation.” **J. Hydraul. Eng., 125(9), 934–942.**

[32] Yen, B. C. (2002). “Open channel flow resistance.” **J. Hydraul. Eng., 128(1), 20–39**.