Study of Hydraulic Characteristics in an Open Channel Flow with vegetation

A Thesis Submitted in Partial Fulfillment of the Requirements for the Degree of

Master of Technology
In
Civil Engineering

KALAGARA PHANINDRA

DEPARTMENT OF CIVIL ENGINEERING

NATIONAL INSTITUTE OF TECHNOLOGY, ROURKELA

2015

Study of Hydraulic Characteristics

in an Open Channel Flow with vegetation

A Thesis Submitted in Partial Fulfillment of the Requirements for the Degree of

Master of Technology
in
Civil Engineering

WITH SPECIALIZATION IN

WATER RESOURCES ENGINEERING

Under the guidance and supervision of

Prof K. K. Khatua

Submitted By:

KALAGARA PHANINDRA

(ROLL NO: 213CE4096)

DEPARTMENT OF CIVIL ENGINEERING

NATIONAL INSTITUTE OF TECHNOLOGY, ROURKELA

2015

DEPARTMENT OF CIVIL ENGINEERING

NATIONAL INSTITUTE OF TECHNOLOGY, ROURKELA

DECLARATION

I hereby declare that this submission is my own work and that, to the best of my knowledge and belief, it contains no material previously published or written by another person nor material which to a substantial extent has been accepted for the award of any other degree or diploma of the university or other institute of higher learning, except where due acknowledgement has been made in the text.

KALAGARA PHANINDRA

NATIONAL INSTITUTE OF TECHNOLOGY
ROURKELA

CERTIFICATE

This is to certify that the thesis entitled, **"Study of Hydraulic characteristics in an open channel flow with vegetation"** submitted by **KALAGARA PHANINDRA** in partial fulfillment of the requirements for the award of Master of Technology Degree in **CIVIL ENGINEERING** with specialization in **"WATER RESOURCE ENGINEERING"** at the National Institute of Technology, Rourkela is an authentic work carried out by him under my supervision and guidance.

To the best of my knowledge, the matter embodied in the thesis has not been submitted to any other University / Institute for the award of any Degree or Diploma.

Date:

Place:

Prof. K.K. Khatua
Dept. of Civil Engineering
National Institute of Technology
Rourkela-769008

ACKNOWLEDGEMENT

I express my sincere gratitude and sincere thanks to **Prof. K.K. Khatua** for his guidance, constant encouragement and support during the course of my Research work. I truly appreciate and value his esteemed guidance and encouragement from the beginning to the end of this works, his knowledge and accompany at the time of crisis remembered lifelong.

I sincerely thank to our Director **Prof. S. K. Sarangi**, and all the authorities of the institute for providing nice academic environment and other facilities in the NIT campus, I express my sincere thanks to Professor of water resource group**, Prof. K .C Patra**, **Prof. A.Kumar** and **Prof R. Jha** for their useful discussion, suggestions and continuous encouragement and motivation. Also I would like to thanks all Professors of Civil Engineering Department who are directly and indirectly helped us.

I am also thankful to all the staff members of Water Resource Engineering Laboratory for their assistance and co-operation during the course of experimental works. I would like to thank my Parents, who taught me the value of hard work by their own example.

At last but not the least, I thank to all those who are directly or indirectly associated in completion of this Research work

Date:

Place:

Kalagara Phanindra

ABSTRACT

Experiments have been conducted in an open channel with rigid vegetation for unsubmerged and submerged flows. Cylindrical rods are used to represent the rigid vegetation. The results show that as flow depth changes the roughness constant also changes. The variation of vertical velocity profiles, depth averaged velocity with the flow depth has also been discussed. The flow resistance though the channel is best expressed with depth averaged velocity. Vertical profiles of point velocity for a vegetative channel from position to position are different as compared to that of a simple channel. The velocity profile patterns are investigated.

The numerical method is applied to calculate velocity profiles in an open channel with vegetation. The results of numerical simulation are compared with the experimental data and numerical results are little over estimating than experimental. In this thesis, a computational fluid dynamics CFD model ANSYS FLUENT is used to simulate for a vegetated open channel flow. The simulation of channel is done by taking k-epsilon model and volume of fluid VOF.

Key Words:

CFD simulation, Vegetated open channel, cylindrical rods, K-epsilon model, Velocity profiles, VOF method.

TABLE OF CONTENTS

LIST OF FIGURES

NOTATIONS

ADV	Acoustic Doppler Velocimetry
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

INTRODUCTION

1.1 OPEN CHANNEL FLOW:

An open channel is defined as a passage in which liquid flows under gravity with free surface i.e., atmospheric pressure acting on the surface of liquid.

Some figures showing open channel flow

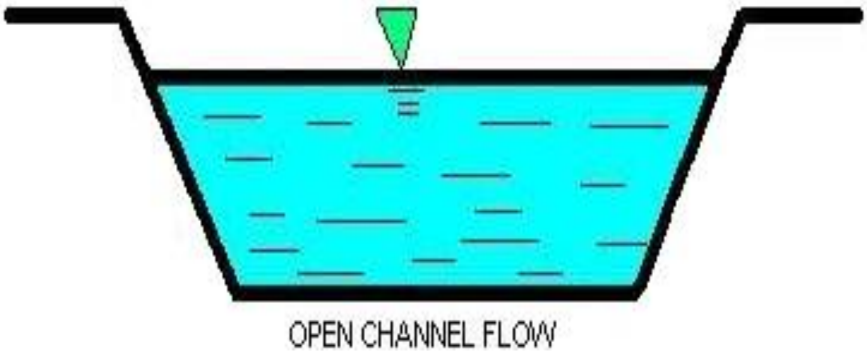


Figure 1.1 OPEN CHANNEL FLOW

1.2 VEGETATED OPEN CHANNEL FLOW:

Vegetation:

Vegetation is defined as a kind of surface roughness, which also reduces the capacity of the channel and retards the flow.

Types of vegetation:

When vegetation is used for waterways it is divided into two types:

I. Natural vegetation i.e. naturally found on the beds and river banks

II. Artificial vegetation i.e. artificially planted

Natural vegetation in rivers and floodplains typically comprises of trees, herbs, shrubs, hedges, bushes and grasses etc.

Figure 1.2 Natural vegetation

Figure 1.3 Artificial vegetation

The vegetation is a very important factor in a channel for evaluation of roughness because it influences the flow across the channel.

The degree of influence in a vegetative channel further depends on the other vegetation characteristics such as:

a. vegetation species
b. distribution
c. flexibility
d. degree of submergence
e. vegetation density
f. vegetation height

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.

1.3 ROUGHNESS COEFFICIENT:

For calculating flow velocity, Manning's formula is generally used. It is widely used in vegetated waterways to solve in relate hydraulic applications. The formula states:

V=K/n Rh2/3 S1/2

Where,

V is the cross-sectional average velocity (L/T; m/s);

K is a conversion factor of (L1/3/T), 1 m1/3/s for SI unit);

n is the Manning's roughness coefficient;

Rh is the hydraulic radius and,

S is the slope of the hydraulic grade line (= hf /L), which is the same as the channel bed slope when the water depth is constant and hf is called difference of hydraulic head across a cross section of channel of length L.

Manning's roughness coefficient is found to vary largely in a vegetated waterway. It depends on depth of flow in the channel.

1.4 NUMERICAL SIMULATION

Computational Fluid Dynamics (CFD) is a computer based numerical analysis tool. Numerical hydraulic model CFD reduces the costs when compared to the experimental model, so there is increase in use of CFD simulation. The basic principle in CFD is applying boundary conditions to the channel and analysing fluid flow by solving equations.

In the present work, the velocity profiles at a section of an Open channel having vegetation are investigated by using a CFD model FLUENT. The results obtained in the experimental channel using ADV are compared with the numerical model. The results of CFD mainly depend on mesh quality, boundary conditions and model taken.

1.5 ADVANTAGES OF NUMERICAL SIMULATION

In experiments we can analyse the flow using the results but full scale readings cannot be obtained, it may be due to instrumental limitations, and at all points we may not be able to take the readings. The three dimensional values may not be exactly calculated. Numerical simulation can be used to overcome all these things. In this it gives readings at all points and can be easily analysed. The main advantage of numerical simulation is it reduces a lot of labour when compared to experimental works.

1.6 OBJECTIVE OF PRESENT STUDY

Velocity profiles in an open channel with vegetation are studied in the present work. The velocity profiles in a channel depend on various parameters like flow depth, vegetation density and height. Out of these parameters flow depth is considered here for estimation of velocity profiles. From literature review it is concluded that very less work has been done using CFD models to validate in vegetated Open channel.

The objectives of the present study are:

- To conduct experiments in a straight open channel flow with vegetation to study (a) Roughness coefficient (b) Velocity distribution (c) Stage discharge.
- To study the impact of flow depth on velocity distribution profiles in the channel.
- To use a CFD model FLUENT to simulate with the experimental vegetated channel and to produce results on velocity distribution.
- Validation and verification of the CFD results such as vertical velocity profiles with the experimental results.

1.7 ORGANISATION OF THESIS

This thesis has seven chapters. In Chapter 1 general *introduction*, in Chapter 2 *literature survey*, in Chapter 3 *experimental work*, in Chapter 4 *numerical modelling*, in Chapter 5 *verification of numerical model* and *discussion of result*, the *conclusions* in chapter 6 and *references* are presented in Chapter 7.

In chapter 1 brief introduction on open channel flow, vegetated open channel and roughness coefficient is described. Numerical simulation is also discussed in this chapter.

In chapter 2 works done by the researchers previously on vegetated open channel is shown.

In chapter 3 details of the experimental channel and arrangements of the channel to do the experiment is shown. Details of the procedure for calculating the velocity values are also shown.

In chapter 4 numerical simulations is explained. Firstly the creation of geometry and meshing is explained. Next the boundary conditions imposed and model used to simulate are described.

In chapter 5 results in the experimental channel are shown and the results are compared with the numerical model.

In chapter 6 conclusions are derived from the experimental and numerical simulation and in chapter 7 references used in this thesis are presented.

LITERATURE REVIEW

2.1 OVERVIEW

The previous work done by other researchers in the field of open channel flow with vegetation which is relevant to the current work is outlined in this chapter. Flow velocity is one of the important aspects in open channel flows. It directly relates to several flow features. The velocity distribution in an open channel flow with vegetation is generally affected by various factors such as vegetation density, pattern, channel roughness which have been critically studied by many renowned researchers. There are several studies found in literature related to vegetated open channel flows.

2.2 PREVIOUS EXPERIMENTAL RESEARCH ON VEGETATED OPEN CHANNEL

Changjun Zhu, Wenlong Hao, and Xiangping Chang (2014) experimentally determined the effects of rigid vegetation on the flow characteristics, where the vegetations were represented by rigid cylindrical rod. Flow field is calculated under the conditions of submerged rigid rods in channel with single layer and double layer vegetations. Experiments were done for various rigid rods spacing’s. The rigid vegetation models were collaborated with the approaching flow in a rectangular channel. Vertical distributions of time-averaged velocity at various stream wise distances were calculated using an acoustic Doppler velocimeter (ADV). The results showed that, in submerged conditions, it is difficult to show the velocity distribution along entire depth using unified function.

Wang Wen, Huai Wen-xin, Gao Meng (2014) described flow structure through the rigid vegetation by RNG k-ε model by comparing with an experimental measurement. By comparing the flow velocity of numerical results versus that of the experimental results, the calculation method was verified. The calculation results showed that the flow structure near the region of short and tall cylinders can hardly be achieved by the experimental calculations. Near top of flow the velocity distribution can be greatly affected by free surface. Due to differences in flow velocity, the hinder effect of tall vegetation is higher than that of short vegetation. The flow values is compared by using the commercial software FLUENT with renormalization group (RNG) *k-epsilon* models.

W.X. Huai, Y.H. Zeng, Z.G. Xu, Z.H. Yang (2009) calculated the vertical velocity distribution in a submerged vegetative open channel flow. The vertical velocity and turbulence behaviour of steady uniform flow with fully submerged artificial rigid vegetation was measured using a 3D Micro ADV, and the vertical velocity distribution and Reynolds shear stress at various vegetation height, vegetation density and measuring positions were obtained. The results indicate that the vertical velocity profile consists of three hydrodynamic regimes (i.e. the upper non-vegetated layer, the outer and bottom layer within vegetation); accordingly various methods had been taken to describe the vertical velocity distribution.

Gregory V. Wilkerson (2006) conducted a study in which wood dowels were used to represent rigid vegetation. The dowel configurations used in the channel were taken to simulate the effects of willow post systems i.e., collections of rigid cylinders placed along a stream bank to reduce stream bank erosion. In addition to it, an analytical model is shown for predicting depth-averaged velocity distributions in straight trapezoidal or rectangular channels with newly constructed willow post systems. The analytical model is founded on wake theory and is applicable to channels with submerged and unsubmerged rigid cylinders. Data from three independent physical model studies were used to validate the analytical model. Depth-averaged velocities predicted using the analytical model, *U*pred, were verified to velocities observed in the physical models, *U*obsd, and yielded discrepancy ratios, *U*pred /*U*obsd, that were typically between 0.80 and 1.20. Results from this study are that significant variables for reducing local velocities are cylinder height, diameter, and density and the arrangement of the cylinders.

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.

EXPERIMENTAL SETUP AND PROCEDURE

3.1. GENERAL

The present research work utilises the flume facility available in the Fluid Mechanics and Hydraulic Engineering Laboratory of the Civil Engineering Department at the National Institute of Technology, Rourkela, India. The basic objective behind these experiments is to get better understanding on the variation of velocity distribution. The following section gives a brief overview of details of hydraulic and geometric parameters of the present vegetated open channel, experimental arrangements, measuring equipments and procedure used in the experimentation process.

3.2 EXPERIMENTAL ARRANGEMENTS

3.2.1 Geometry Setup

Experiments were conducted in simple rectangular channel with cross section having dimension 12m×0.6m×0.45m. The whole channel was fabricated by using 19 mm thick water resistant plywood in bed. 6.5 mm diameter iron rods each raised to a height of 10 cm were inserted into the plywood in a staggered pattern having a spacing of 10 cm. The test reach in the channel is 6m i.e., length of the channel with vegetation.

Water will be supplied through centrifugal pumps (15 hp) discharging into a overhead tank. Pump delivers water from the overhead tank to the flume with the maximum discharge of 0.047 m^3/s. Water was supplied to the flume from an underground reservoir via an overhead tank by centrifugal pump.

Water entered the channel via an upstream rectangular notch specifically built to measure actual discharge in the laboratory channel. An adjustable vertical gate along with flow straightners was provided in upstream section to reduce turbulence and velocity of approach in the flow near the notch section. In the downstream end there will be a measuring tank followed by a sump which will feed to over head tank through pumping, thus completing recirculation.

At the downstream end another adjustable tail gate was provided to control the flow depth and maintain a uniform flow in the channel. A movable bridge was provided across the flume for both span wise and stream wise movements over the channel area so that each location on the plan of simple channel could be accessed for taking measurements.

The aspect ratio is $\delta = 1.33$.

Slope of the channel bed is **0.00736.**

The discharge varied between **0.002345 m^3/s to 0.013564 m^3/s.**

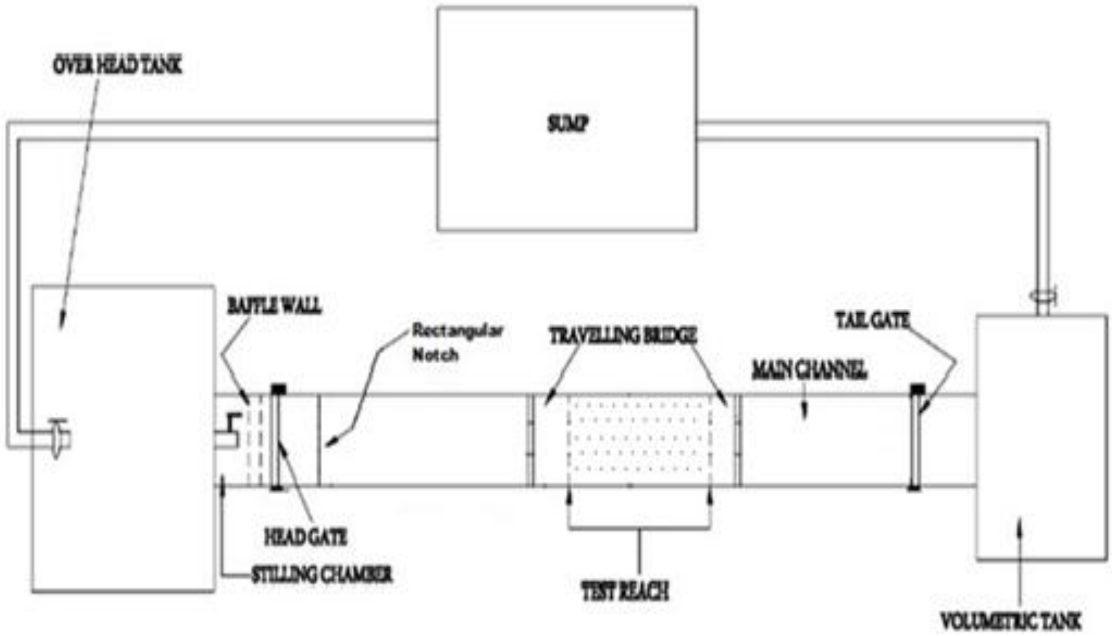


Fig 3.1 plan view of experimental setup of the channel

3.3 PROCEDURE

In the experiments velocity was measured by using ADV. The velocity was measured in two conditions one is submerged and another is unsubmerged. ADV calculates the velocity in three dimensions.

The most important thing is to choose the section to place ADV in the channel to calculate the velocity values. Section is chosen by using two criteria. First thing is development length. It is found in such a way that the channel average velocity (V=Q/Bh) should coincide with the velocity of the section. The velocity of section is calculated by taking two points in a section and calculating the average velocity value in that section.

Second thing is the section covers the maximum cylinders so that it should cover the maximum points as shown in fig.

Discharge is calculated at the end of the tank using glass tube indicator and volumetric tank. Water falls into volumetric tank from outlet and the rise in water is noted by glass tube indicator. The time for rise in glass tube indicator is noted using stop watch. The time is noted for every 1cm rise of water.

Now discharge =area of volumetric tank * rise in level/time. Area of volumetric tank is 20.93square mt.

The experiment is done in two depths one is submerged and another is unsubmerged.

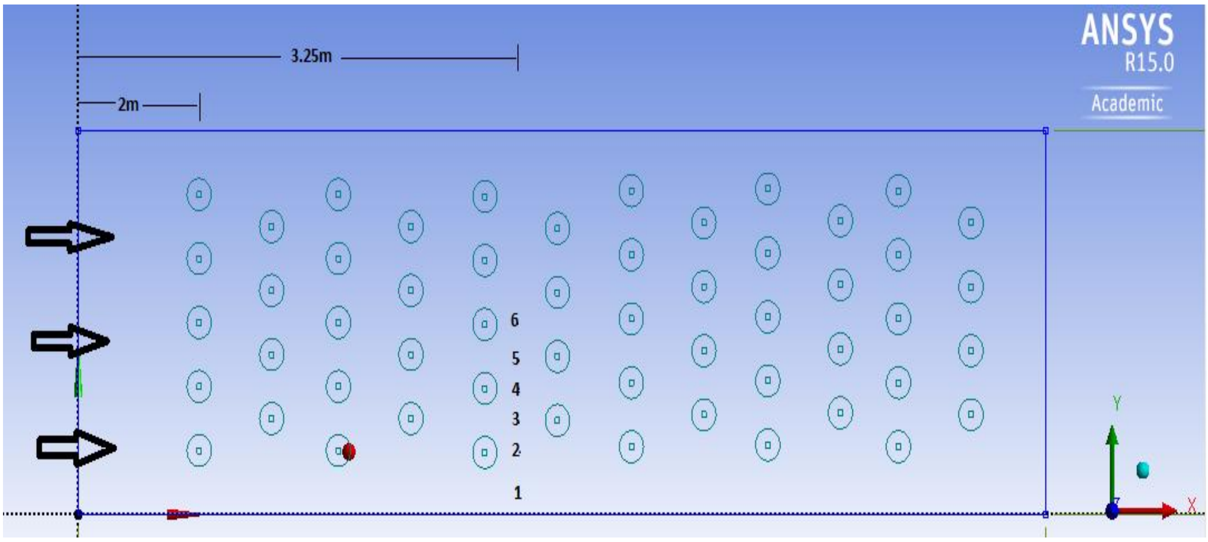


Fig 3.2 top view of the channel showing the section and points where readings are taken

Fig 3.3 photo of vegetated open channel

Fig 3.4 channel with unsubmerged flow

Fig 3.5 channel with submerged flow

Fig 3.6 front view of overhead tank

Fig 3.7 photo of volumetric tank

Fig 3.8 photo of channel inlet along with Straightners

Fig 3.9 photo of channel outlet

Fig 3.10 ADV setup in the channel

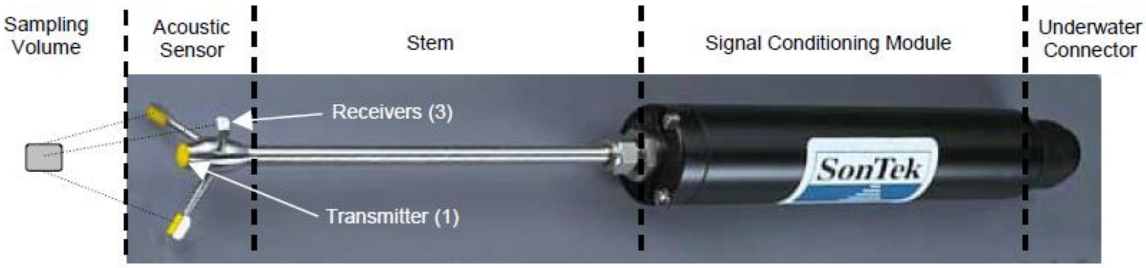


Fig 3.11 Details of ADV parts

NUMERICAL SIMULATION

4.1 DESCRIPTION OF NUMERICAL MODEL PARAMETERS

In this study, for numerical simulation a computational fluid dynamics CFD model FLUENT is used. The various parameters used in this simulation is k-epsilon model, volume of fluid VOF and height of liquid HOL. The model uses transient flow for simulation of the vegetated channel. In addition to this a non iterative solution method SIMPLE is used. Because it will converge the flow faster which means the flow do not change with further iterations.

4.2 METHODOLOGY

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.

4.2.1 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.

The simulation was done on an open channel with rigid vegetation. The setup of the vegetated channel is shown in the fig 4.1 and 4.2.

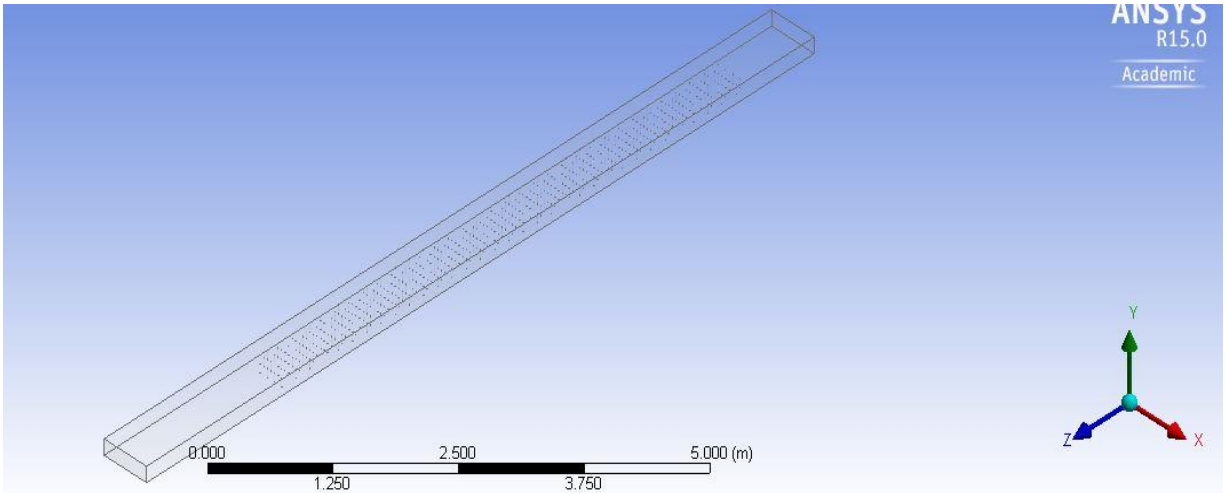


Fig 4.1 geometry of the channel in ANSYS

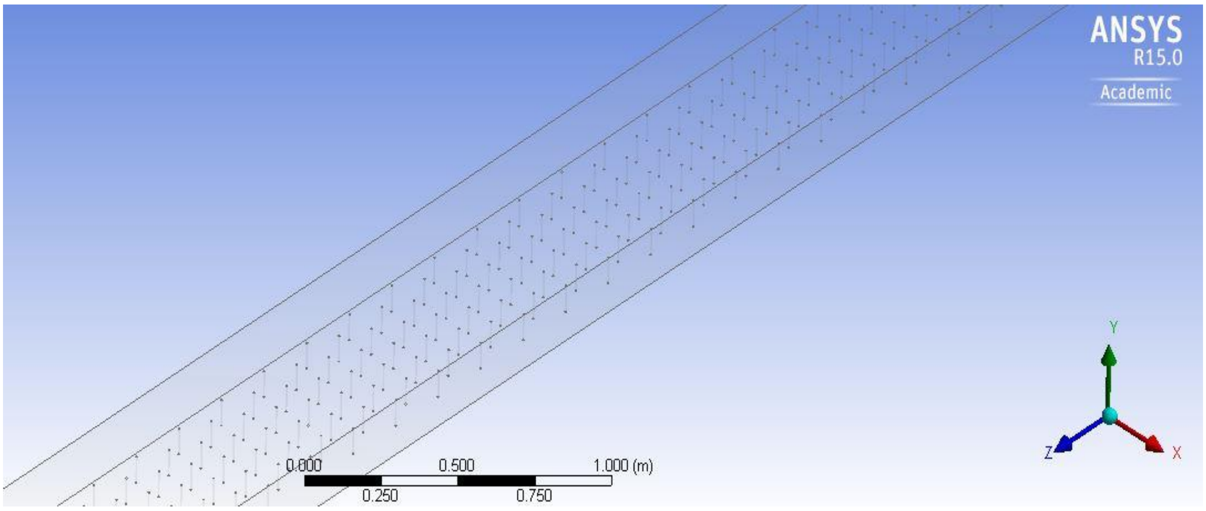


Fig 4.2 close view of geometry showing cylinders

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. Cylinders
4. Sidewalls
5. Bottom surface
6. Top symmetry

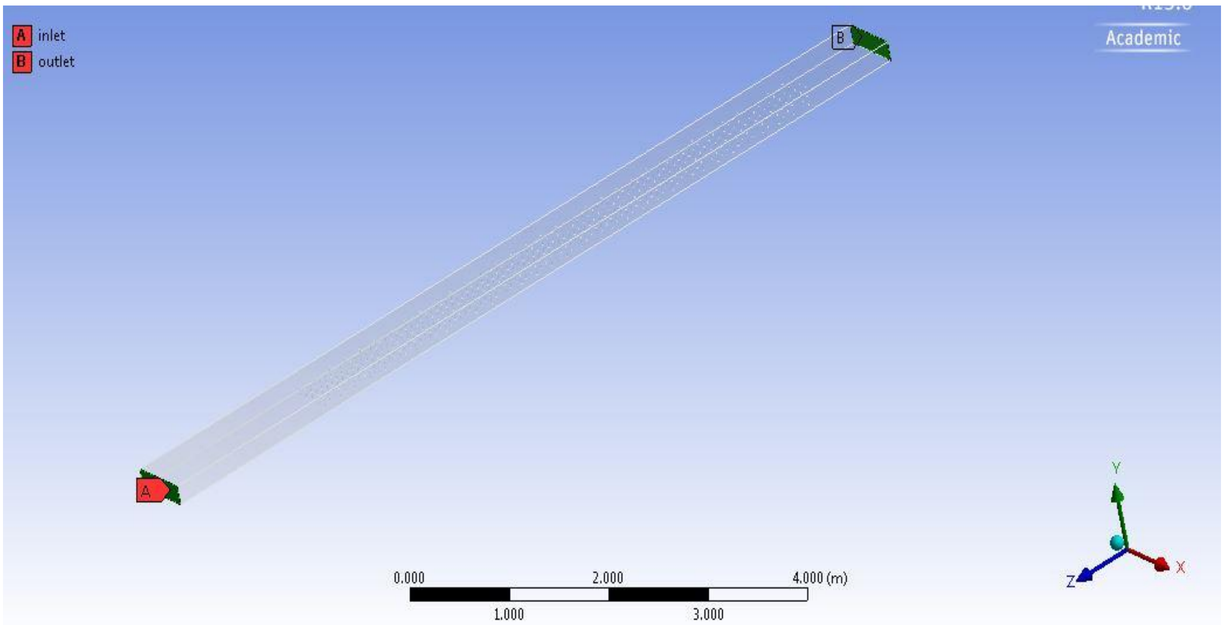


Fig 4.3 inlet and outlet in a vegetated open channel using ANSYS

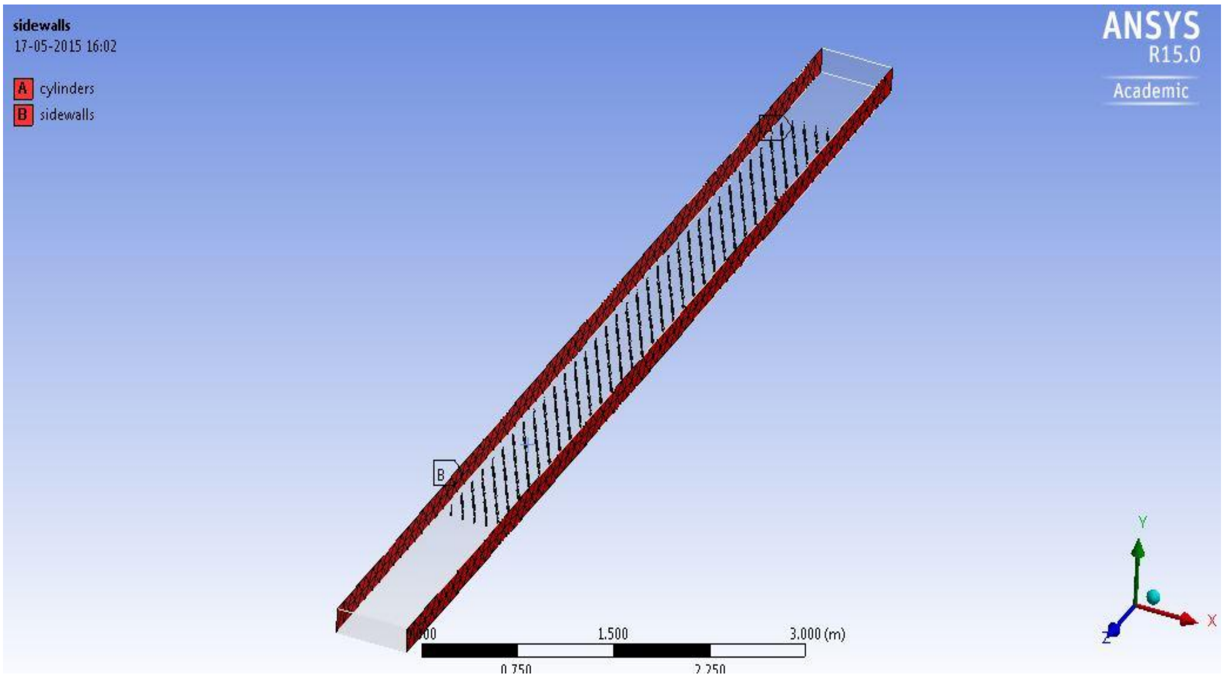


Fig 4.4 cylinders and sidewalls in a vegetated open channel using ANSYS

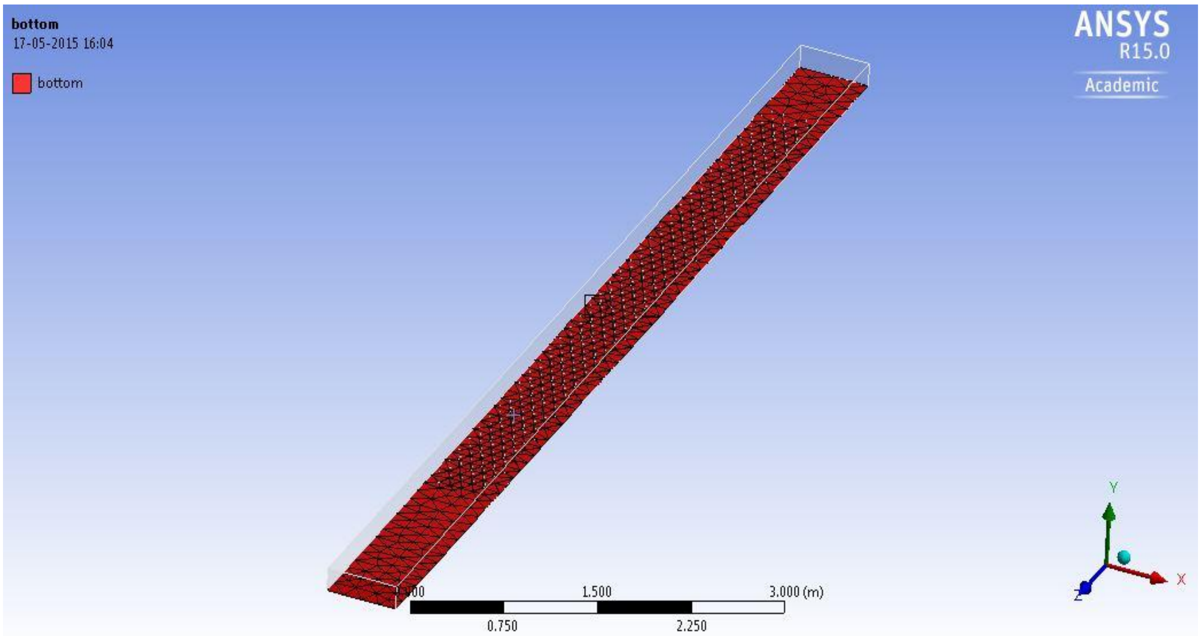


Fig 4.5 bottom of channel using ANSYS

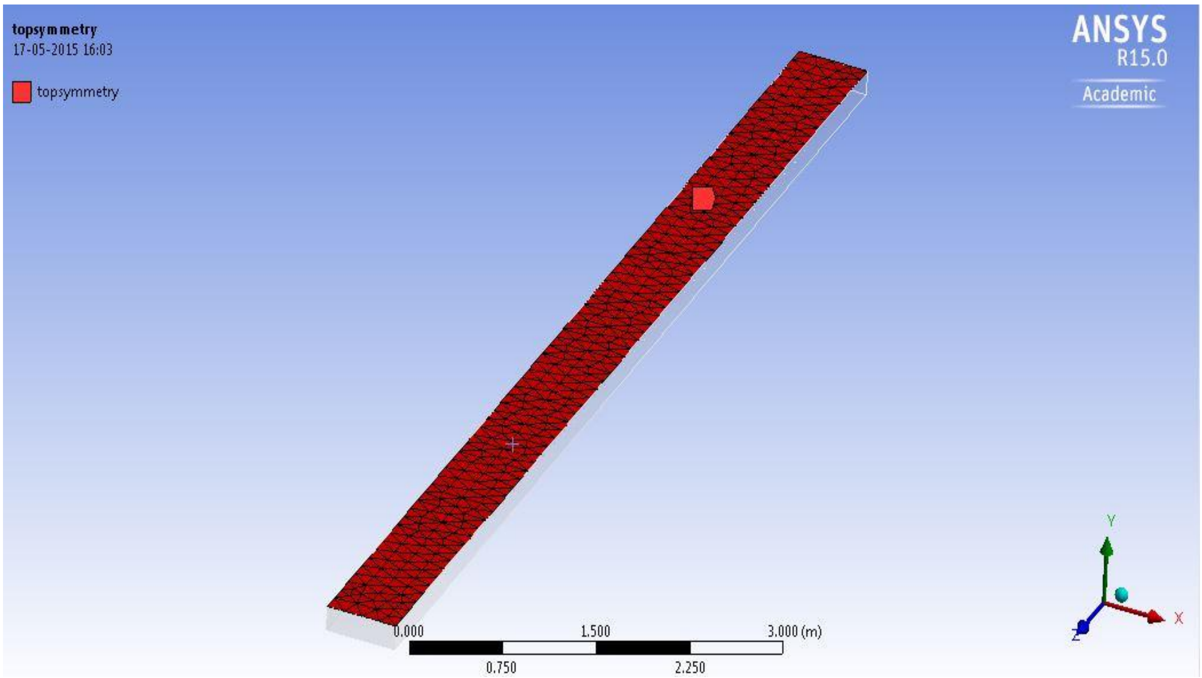


Fig 4.6 top surface of channel using ANSYS

4.2.2 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 4.7.

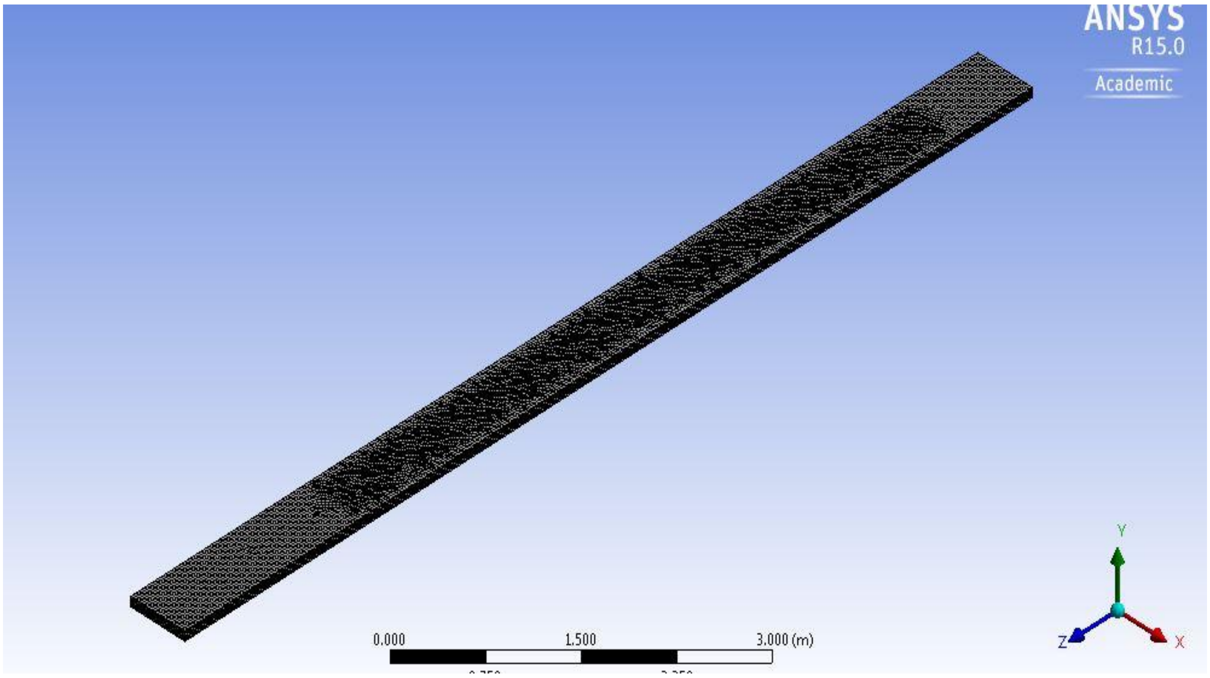


Fig 4.7 detailed meshing of a channel using ANSYS

4.2.3 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).

RNG k-epsilon model:

The RNG model Re-Normalisation Group (RNG) renormalise the Navier-Stokes equations, to account for the effects of smaller scales of motion. There are a number of ways to write the transport equations for k and ϵ, a simple interpretation where bouyancy is neglected is

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

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

Where

$$C^*_{2\epsilon} = C_{2\epsilon} + \frac{C_\mu \eta^3(1 - \eta/\eta_0)}{1 + \beta\eta^3}$$

$$\eta = Sk/\epsilon$$

$$S = (2S_{ij}S_{ij})^{1/2}$$

They are given below with the commonly used values in the standard k-epsilon equation in brackets for comparison:

$C_\mu = 0.0845$ (0.09)

$\sigma_k = 0.7194$ (1.0)

$\sigma_\epsilon = 0.7194$ (1.30)

$C_{\epsilon 1} = 1.42$ (1.44)

$C_{\epsilon 2} = 1.68$ (1.92)

$\eta_0 = 4.38$

Giving boundary conditions to channel is also important. If improper boundary conditions are given the results can be either over estinated or under estimated. Here transient flow is considerd for the simulation.

The effects of gravity and channel slope implemented via a resolved gravity vector. Here the angle θ represents the angle between the bed of the channel and the horizontal, the gravity vector is resolved in x, y and z components as :

$$(0, -\rho g \cos\theta, \rho g \sin\theta)$$

Where θ = angle between bed surface to horizontal axis and tan θ=slope of the channel.

The channel walls i.e. side walls and bottom are represented as non-slip walls. A no-slip boundary condition is the most common boundary condition implemented at the wall and prescribes that the fluid next to the wall assumes the velocity at the wall, which is zero i.e.

For top free surface generally symmetry boundary condition is used. This specifies that the shear stress at the wall is zero and the stream wise and lateral velocities of the fluid near the wall are not retarded by wall friction effects as with a no-slip boundary condition. This condition follows that, no flow of scalar flux occurs across the boundary.

4.2.4 Post-processing

Post processing is the part of fluent where the results are shown after completing the initilisation and calculation. Here the reults shown are

1. velocity contours
2. x-y plots
    (a) vertical velocity profiles
    (b) depth averged velocity

VALIDATION AND VERIFICATION OF RESULTS

5.1 EXPERIMENTAL RESULTS

The experimental results of the velocity profiles, roughness coefficient variation, stage discharge is presented in this chapter. Analysis is done for two flow depths one is unsubmerged and another is submerged. Position of points and section taken were shown in the figure 5.1.

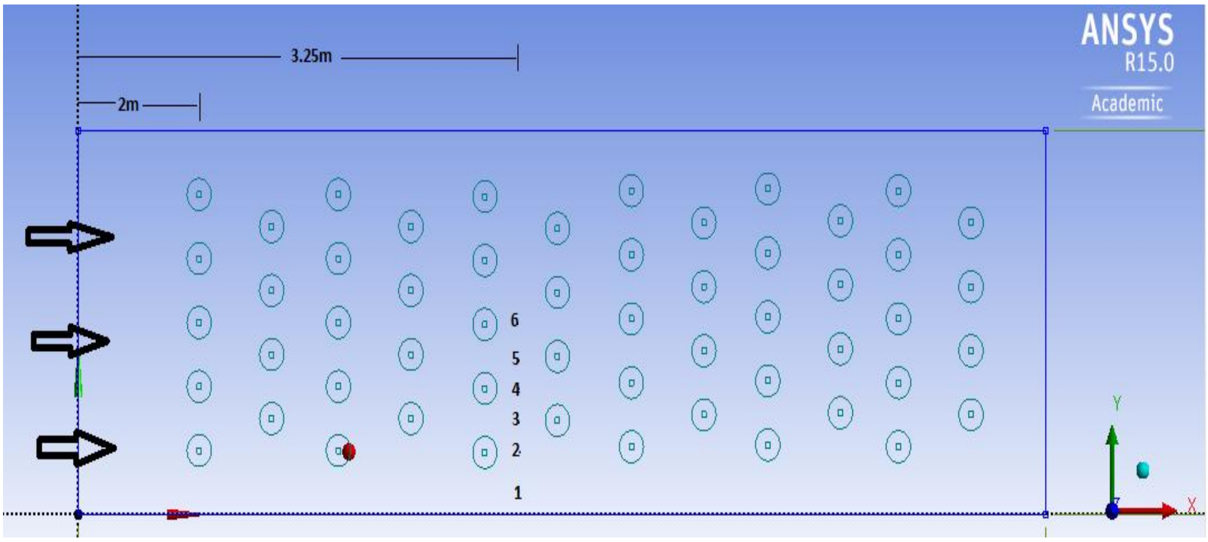


Fig 5.1 top view of the channel showing the section and points where readings are taken

The section is taken 3.25m away from the inlet for both unsubmerged and submerged depths.

Total six points 1,2,3,4,5,6 are taken at a section.

1st point is taken from edge of the channel sidewall and distance from wall is 5cm.

2nd point is 10cm distance from edge of the side wall

3rd point is 15cm distance from edge of the side wall

4th point is 20cm distance from edge of the side wall

5th point is 25cm distance from edge of the side wall

6th point is 30cm distance from edge of the side wall

5.1.1 **RESULTS IN UNSUBMERGED CONDITION**:

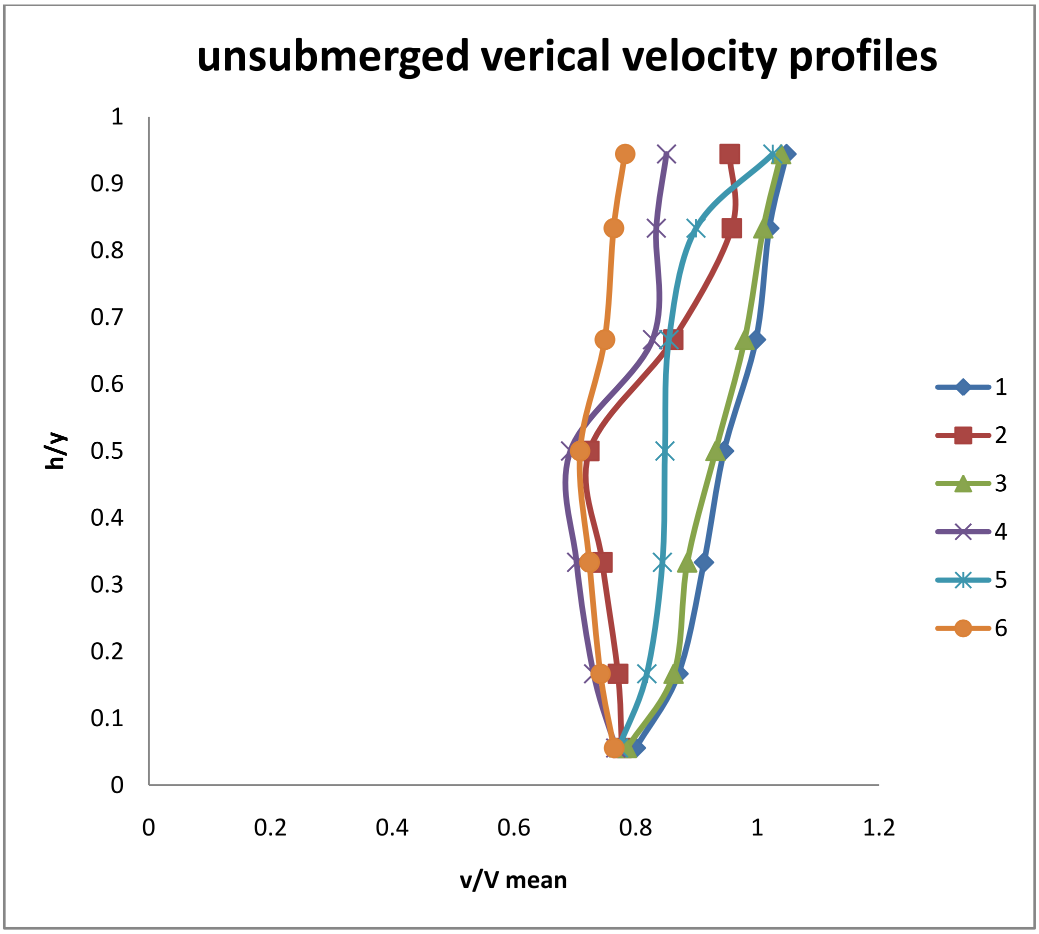


Fig 5.2 vertical velocity profiles of unsubmerged flow

The results shown in the above figure 5.2 is calculated for a depth of 9cm. The position of points 1, 2, 3, 4, 5, 6 are shown in the fig 5.1. The graph is representation of vertical velocity profiles in unsubmerged condition.

The X-axis is showing variation of v/V mean where v=velocity at a particular position in the channel where readings are taken and V mean=apparent channel velocity.

The Y-axis is showing the variation of h/y where h=height of the point from bottom of bed where velocity is calculated and y=depth of water in the channel.

By observing the results it is concluded that the velocity profiles behind the cylinders decreases and increases. The velocity values just go on increasing where there are no cylinders.

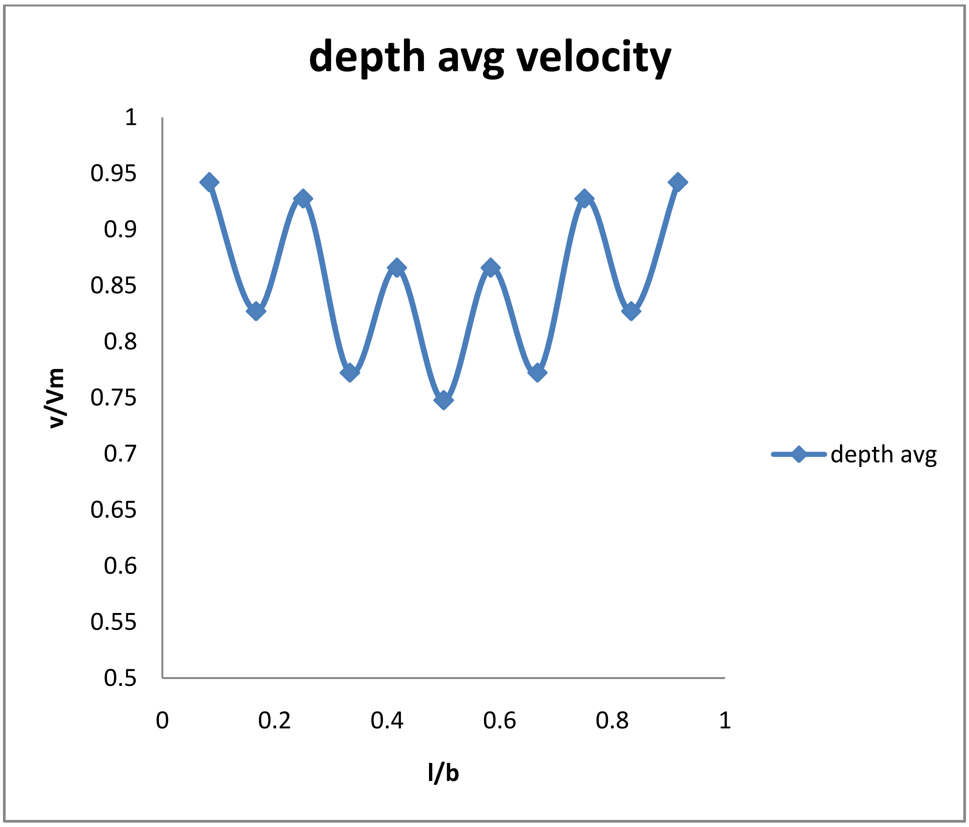


Fig 5.3 depth averaged velocity in unsubmerged flow

The results shown in the above figure 5.3 is calculated for a depth of 9cm. The position of points 1,2,3,4,5,6 are shown in the fig 5.1. The graph is representation of deth averaged velocity profiles in unsubmerged condition.

The X-axis is showing the variation of l/b where l=length of the point from the wall side where velocity is calculated and b=width of the channel.

The Y-axis is showing variation of v/V mean where v=average velocity at a particular position in the channel where readings are taken and V mean=apparent channel velocity.

5.1.2 RESULTS IN SUBMERGED CONDITION:

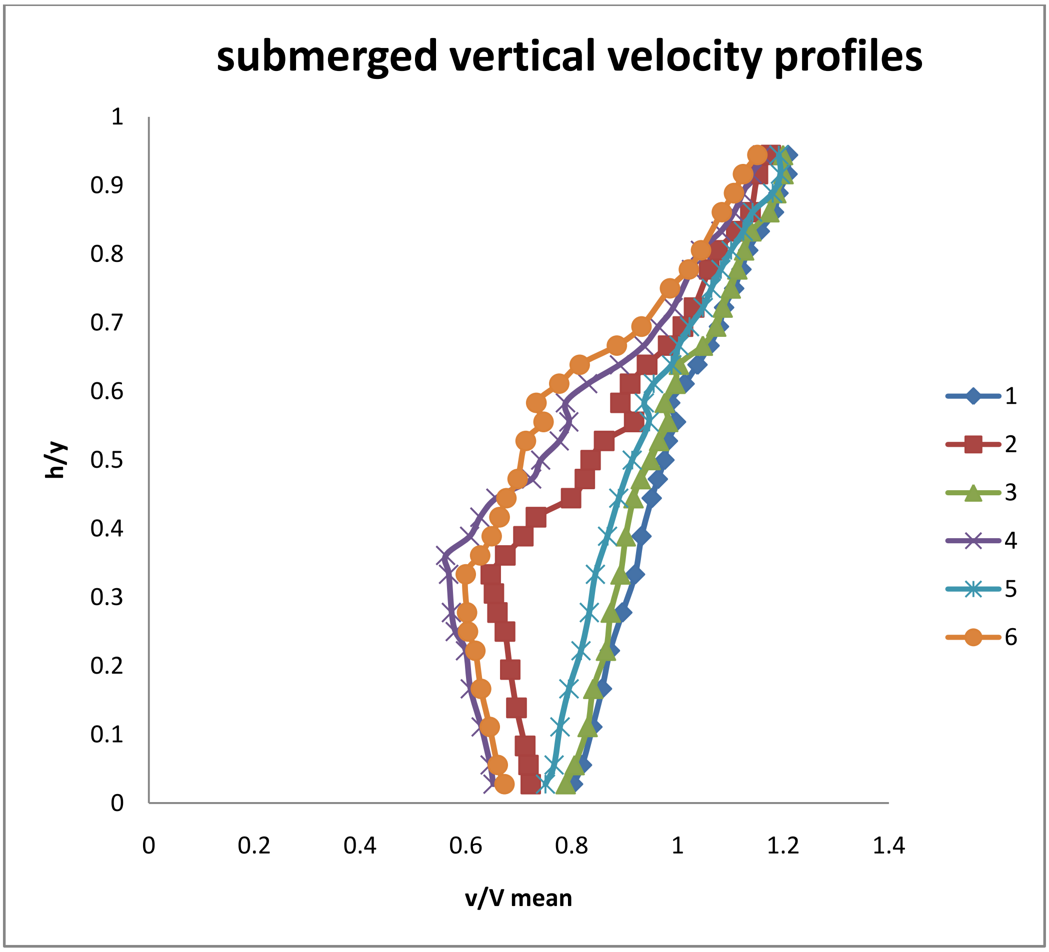


Fig 5.4 vertical velocity profiles of submerged flow

The results shown in the above figure 5.4 is calculated for a depth of 18cm. The position of points 1, 2, 3, 4, 5, 6 are shown in the fig 5.1. The graph is representation of vertical velocity profiles in submerged condition.

The X-axis is showing variation of v/V mean where v=velocity at a particular position in the channel where readings are taken and V mean=apparent channel velocity.

The Y-axis is showing the variation of h/y where h=height of the point from bottom of bed where velocity is calculated and y=depth of water in the channel.

By observing the results it is concluded that the velocity profiles behind the cylinders decreases and increases. The velocity values just go on increasing where there are no cylinders. Above the cylinders that is above 10 cm the velocity value just decrease and increases for all the points.

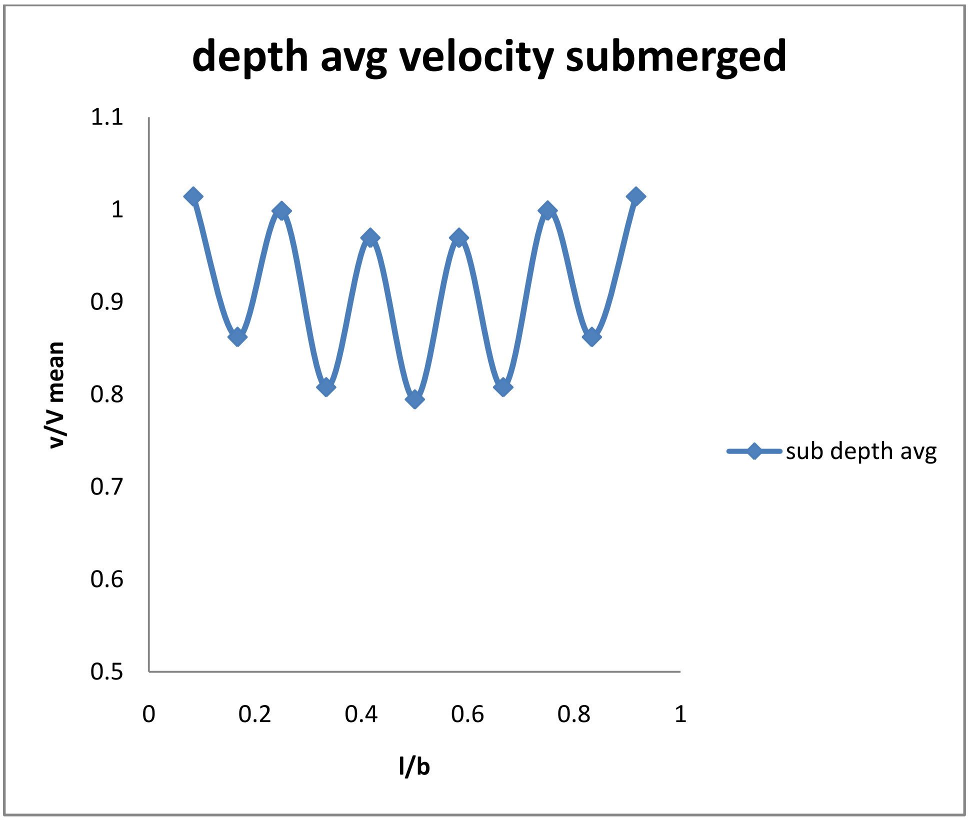


Fig 5.5 depth averaged velocity in submerged flow

The results shown in the above figure 5.5 is calculated for a depth of 18cm. The position of points 1,2,3,4,5,6 are shown in the fig 5.1. The graph is representation of deth averaged velocity profiles in unsubmerged condition.

The X-axis is showing the variation of l/b where l=length of the point from the wall side where velocity is calculated and b=width of the channel.

The Y-axis is showing variation of v/V mean where v=average velocity at a particular position in the channel where readings are taken and V mean=apparent channel velocity.

5.1.3 STAGE DISCHARGE:

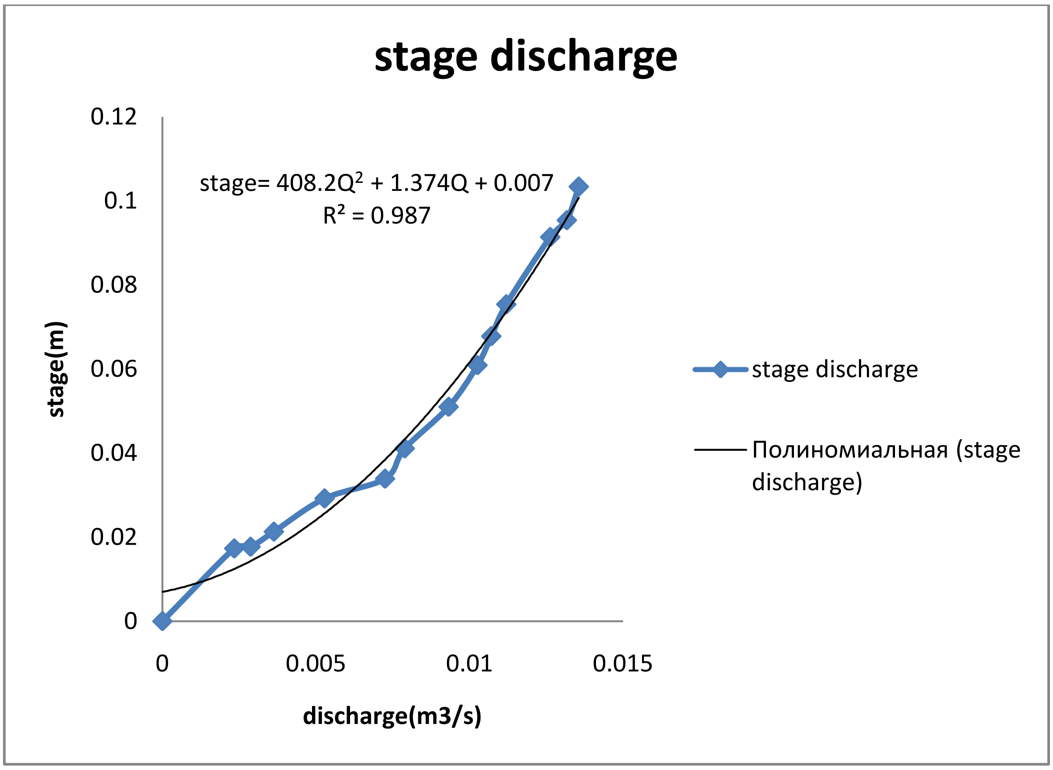


Fig 5.6 stage discharge curve in the channel

The above figure 5.5 shows the stage discharge curve in the vegetated channel. The readings are taken at a distance of 3.25m from the inlet. Discharge is calculated for different depths. As depth increases discharge is also increasing.

On X-axis variation of discharge is taken

and on Y-axis variation of depth is taken.

5.1.4 ROUGHNESS COEFFICIENT VARIATION:

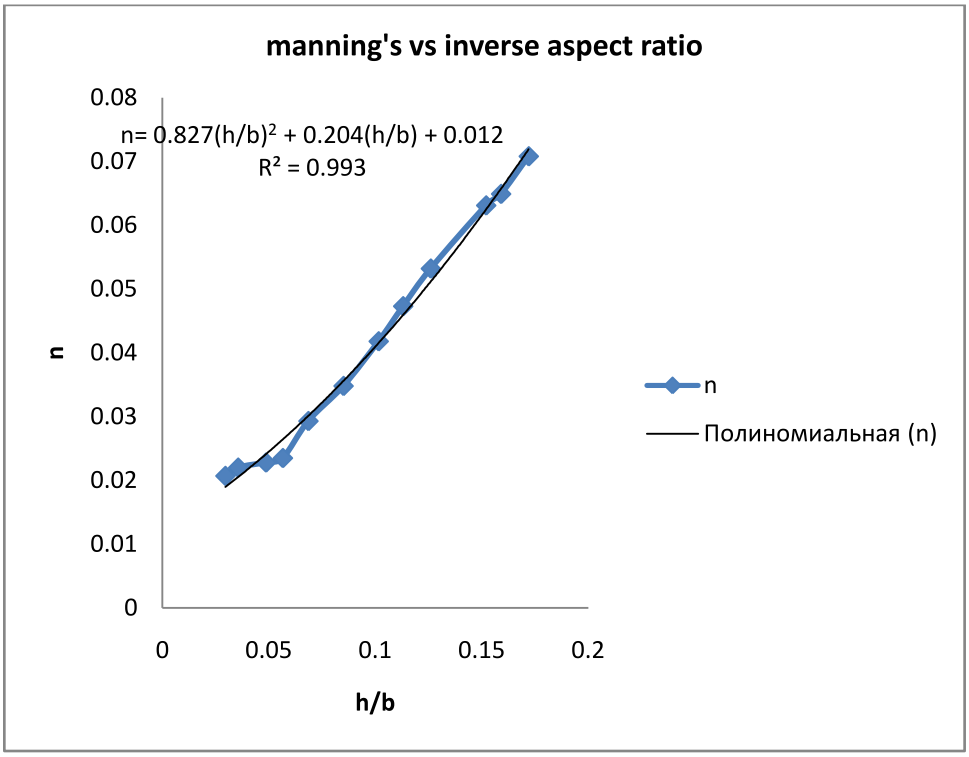


Fig 5.7 manning's coefficient vs inverse aspect ratio in the channel

The above figure 5.6 represents the variation of manning's coefficient and inverse aspect ratio in the vegetated open channel. As the depth of water increases roughness value in the channel is also increasing.

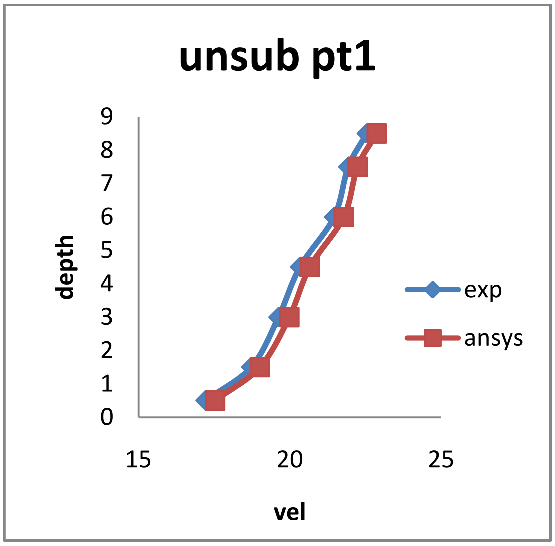
On X-axis inverse aspect ratio h/b is taken where h=height of the point from bottom of the bed where readings of velocity is taken and b= width of the channel.

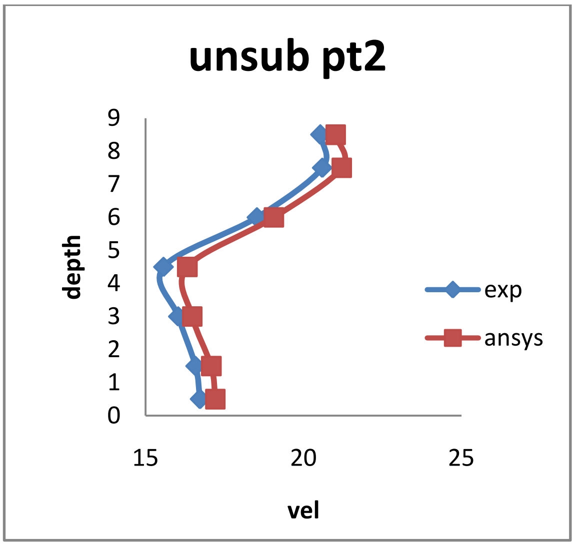
On Y-axis variation of manning's coefficient is taken.

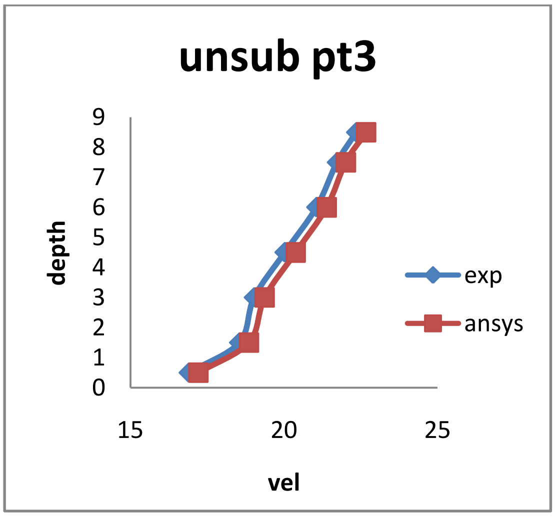
5.2 VALIDATION OF NUMERICAL AND EXPEIMENTAL RESULTS:

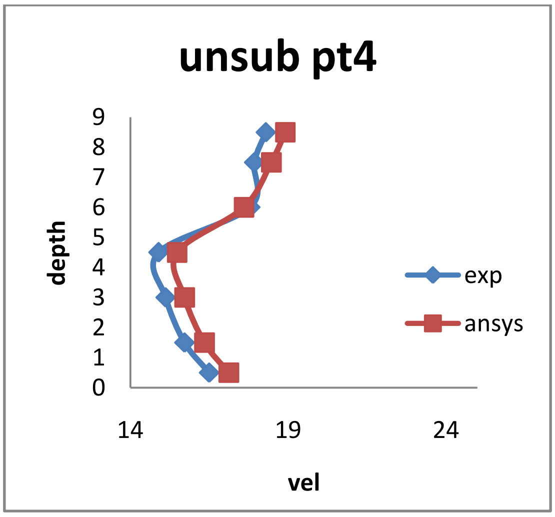
The numerical results are compared with experimental values in this section for both unsubmerged and submerged flow.

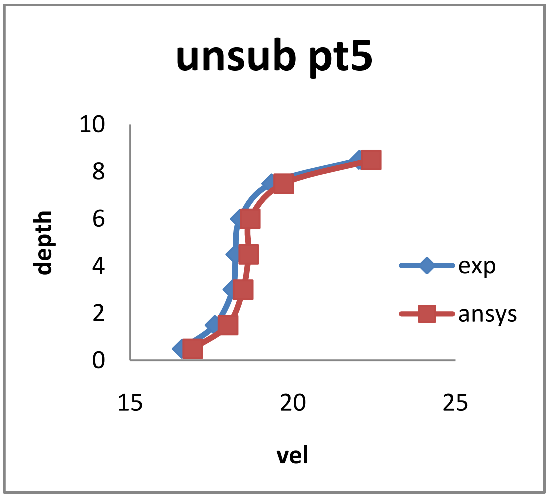
5.2.1 LONGITUDINAL VELOCITY DISTRIBUTION IN THE CHANNEL FOR UNSUBMERGED FLOW:

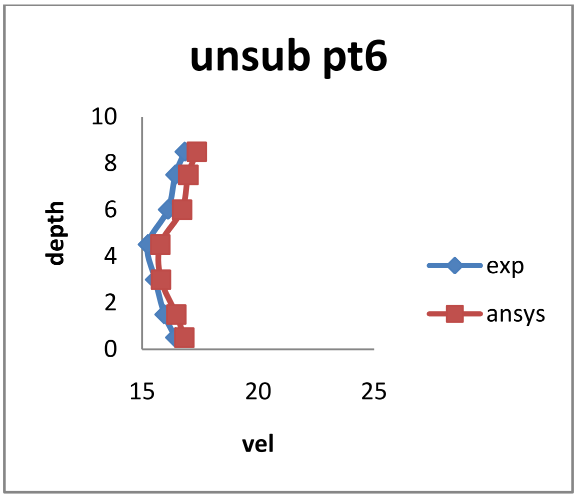




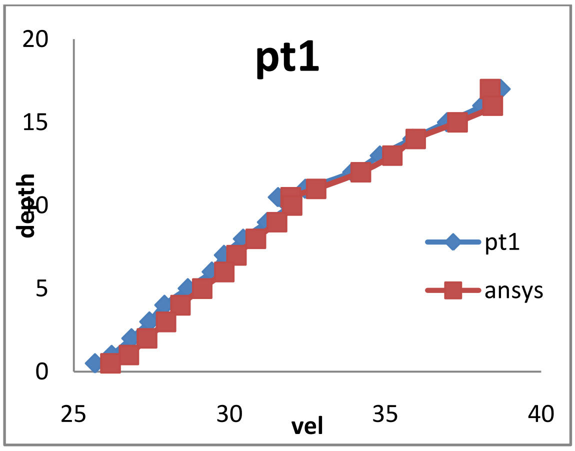


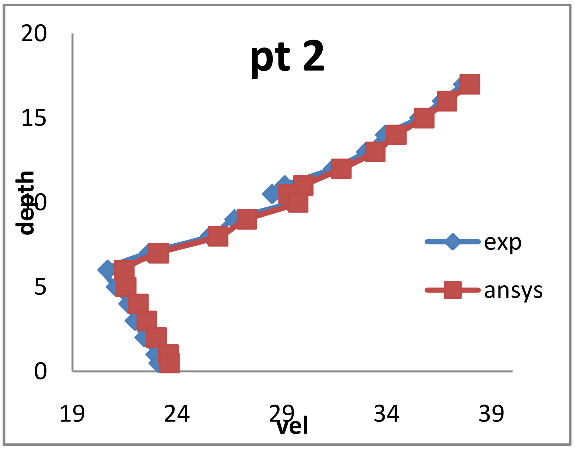


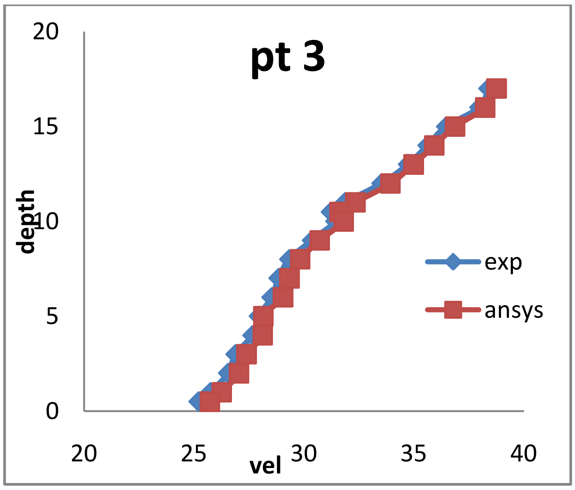


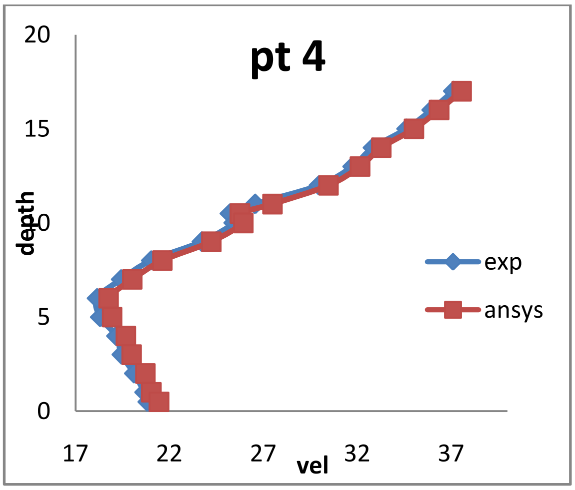


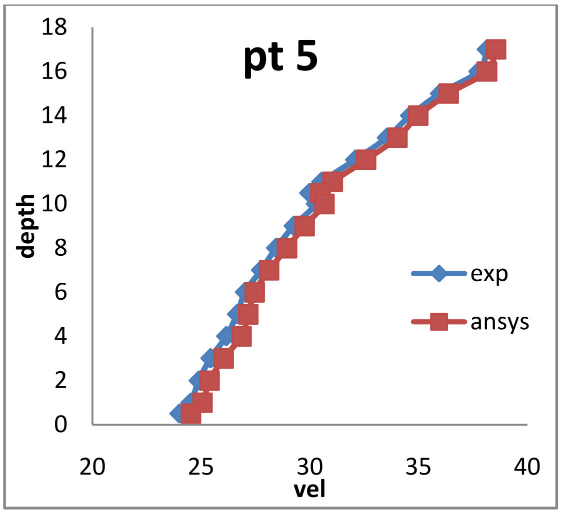
5.2.2 LONGITUDINAL VELOCITY DISTRIBUTION IN THE CHANNEL FOR SUBMERGED FLOW:

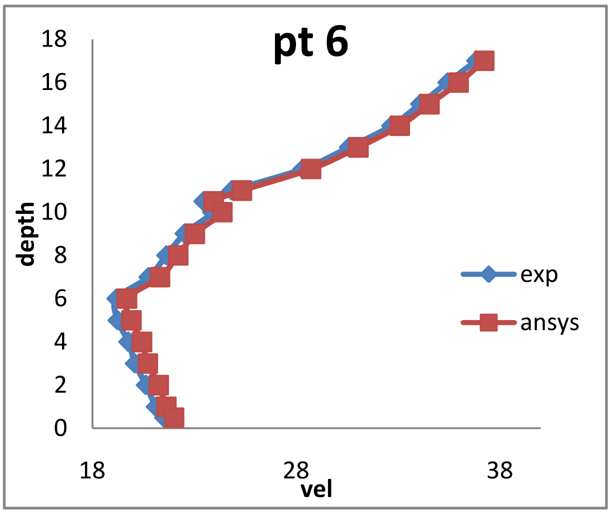




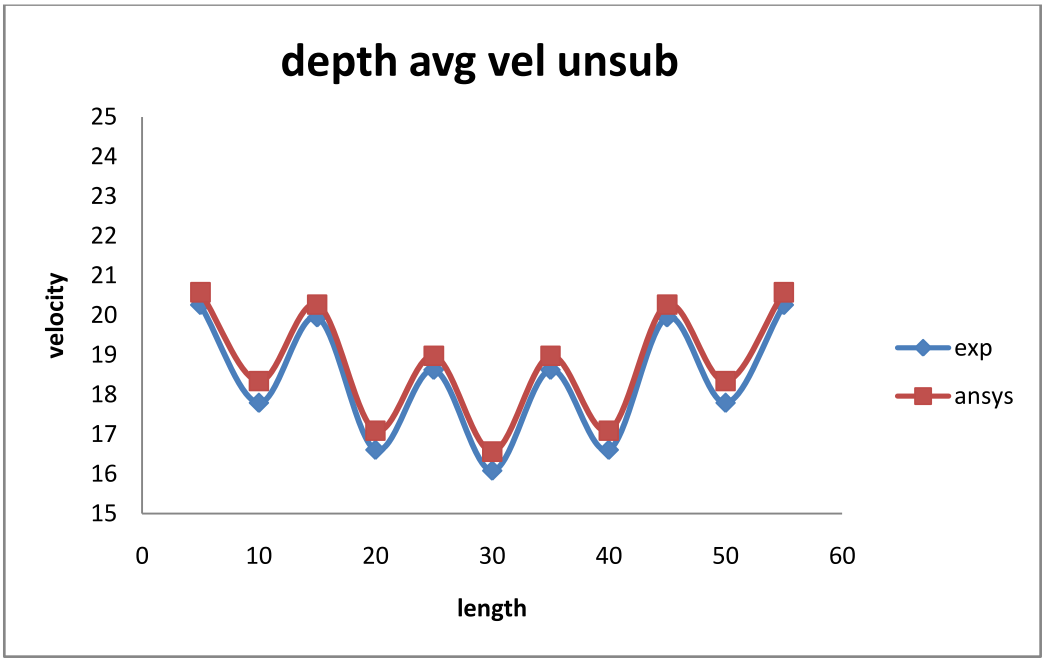




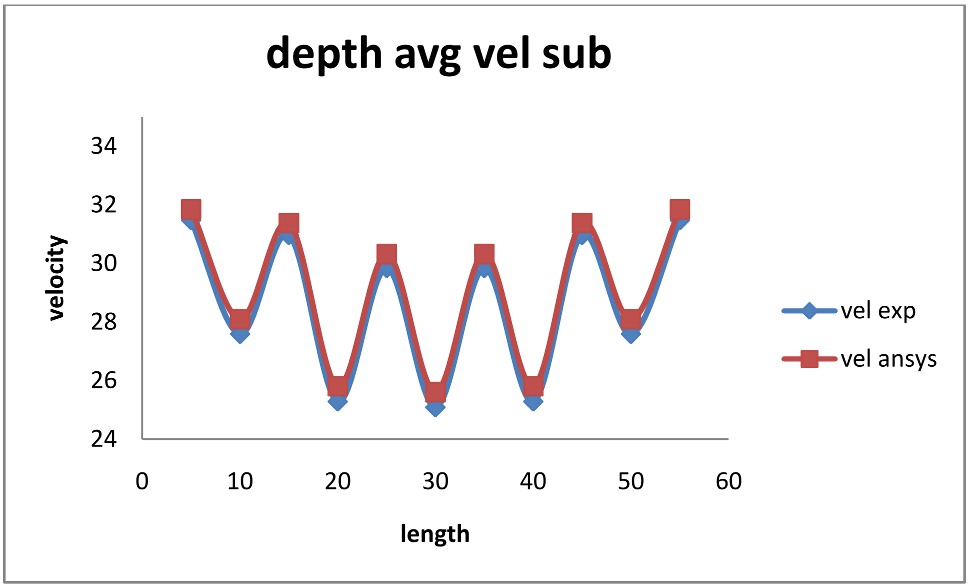




5.2.3 DEPTH AVERAGED VELOCITY IN THE CHANNEL FOR UNSUBMERGED FLOW:



5.2.4 DEPTH AVERAGED VELOCITY IN THE CHANNEL FOR SUBMERGED FLOW:



5.3 CONTOURS OF LONGITUDINAL VELOCITY IN THE EXPERIMENT:

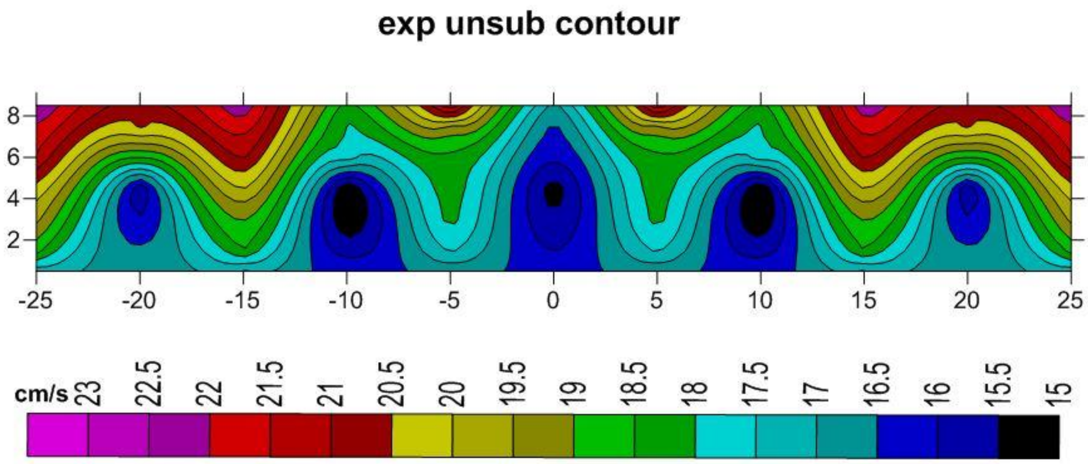


Fig 5.8 contours of longitudinal velocity in unsubmerged flow

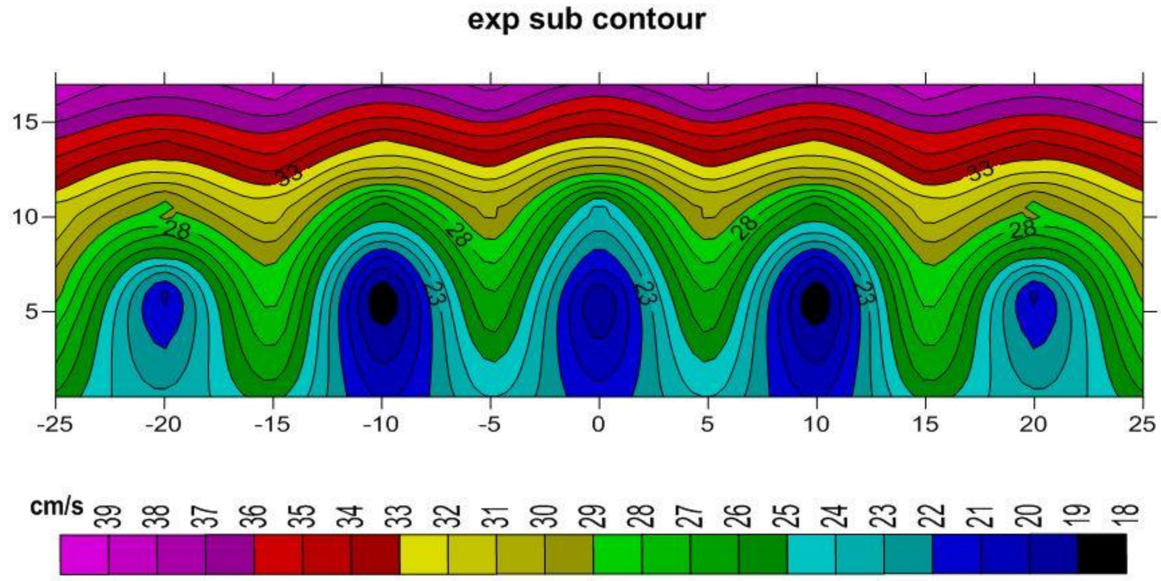


Fig 5.9 contours of longitudinal velocity in submerged flow

5.4 CONTOURS OF LONGITUDINAL VELOCITY IN THE ANSYS:

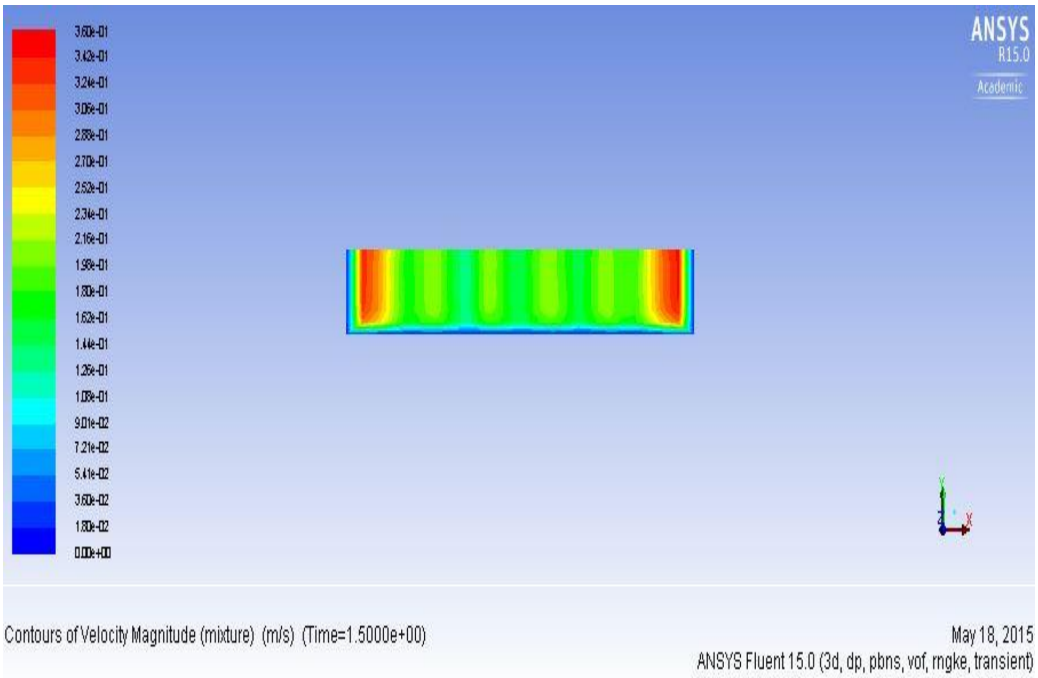


Fig 5.10 contours of longitudinal velocity in unsubmerged flow

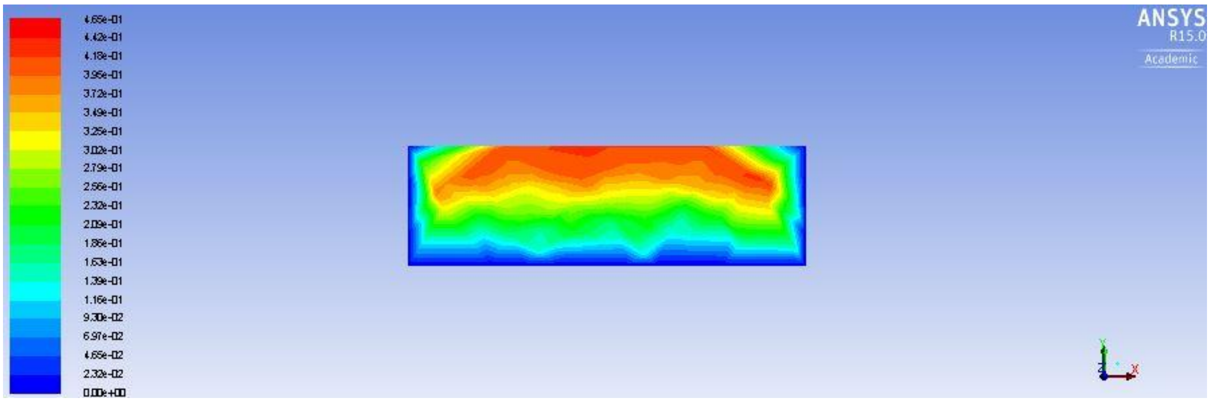


Fig 5.11 contours of longitudinal velocity in submerged flow

CONCLUSIONS AND FURTHER WORK

6.1 CONCLUSIONS

In the present study, the experimental and numerical simulation of an open channel flow with vegetation has been carried out. On the basis of the results such as velocity distribution, depth average velocity distribution along the channel bed for two different depths 9cm and 18cm in a vegetated open channel, the point to point observations are drawn. Conclusions of the work are as follows:

- Numerical simulation in an open channel with rigid vegetation have been successfully verified and validated with experiments.
- Results show that the values of numerical simulation are over estimating than that of experimental values with error less than 5%. The numerical values are little more due to meshing which is insufficient due to academic software which is not giving output if nodes are crossing 5,00,000.
- As the depth of water increases roughness value in the channel is also increasing as shown in Fig 5.6. The reason is as water depth increases the discharge is also increasing due to which velocity and depth also increasing. It is known that manning's coefficient is inversely proportional to velocity and directly proportional to hydraulic radius. But increase in hydraulic radius is more when compared to increase in velocity. Increase in velocity is less due to obstruction created by cylinders.

Longitudinal vertical velocity profiles are calculated in two depths one is unsubmerged and other is submerged. The locations where the velocities are calculated is shown in fig 5.1.

Unsubmerged flow:

- It is seen that in all cases of 2,4,6 the velocity is decreasing while we go from bottom towards the upwards, after mid depth the velocity suddenly increases.
- The increase is more for points going towards the centre. But for 6th point the increase in velocity is less as compared to 2,4. At the centre 6th point the velocity change is very less. Reason is there is much mixing(turbulence) is more bringing the uniformity.
- But for 3 and 5 points which is lying in between the rods without direct obstruction the velocity increases. But 5th point towards the middle of channel is having less velocity when compared to other points 1,3 is due to having more fluctuation.
- The 1st point is having more velocity when compared to all other points are due to there is no obstruction in the whole channel from inlet.

Submerged flow:

- The analysis of longitudinal velocity is carried in two sections. One section is from bottom of channel to 10cm depth which is height of vegetation. Second section is from 10cm to free surface of the water that is 18cm.
- In the first section the reasons for the variation of the longitudinal velocity profiles is same as that of unsubmerged flow.
- In the second section after 10cm the longitudinal velocity decreases a little in the initial and goes on increases till the free surface. In initial there is little decrease due to presence of drag force. After that velocities goes on increasing in all positions as there is no obstruction.

6.2 SCOPE FOR FUTURE WORK

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. Rectangular 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 rectangular channel can be compared with converging, diverging channel and can be applied to the natural channels.

REFERENCES

- 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.*

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

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

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

- 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.*

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

- 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.*

- 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.*

- 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.*

- 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.*

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

- 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.*

- 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.*

- 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.*

- 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.*

- Petryk, S., and Bosmajian, G. (1975). ‘‘Analysis of flow through vegetation.’’*J. Hydraul. Div., 101,HY7!, 871–884.*

- Stoesser, T., Kim, S. J., and Diplas, P. (2010). "Turbulent flow through idealized emergent vegetation." *J. Hydraul. Eng., 136(12), 1003–1017.*

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

- 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.*

- Thompson, G. T., and Roberson, J. A. (1976). ''A theory of flow resistance for vegetated channels.'' *Trans. ASAE, 19~2!, 288–293.*

- Tsihrintzis, V. A. (2001). "Discussion of 'Variation of roughness coefficients for unsubmerged and submerged vegetation'' *J. Hydraul. Eng., 127(3), 241–244.*

- Tsujimoto, T., and Kitamura, T. (1990). ''Velocity profile of flow in vegetated-bed channels.'' *KHL Progressive Rep. No. 1, Kanazawa University, Japan.*

- 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.*

- 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.*

- Yen, B. C. (2002). "Open channel flow resistance." *J. Hydraul. Eng., 128(1), 20–39.*