IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE)
e-ISSN: 2278-1684, p-ISSN: 2320-334X.
PP 07-12
www.iosrjournals.org

CFD Simulation And Two Phase Modeling Of a Compound Meandering Channel

Mamata Rani Mohapatra[1], Abinash Mohanta[2], K.C Patra[3], K. K Khatua[4], Deepika P. Palai

Abstract: *River is the author of its geometry. Almost all natural rivers meander. In fact Straight River reaches longer than 10 to12 times the channel widths are non existence in nature. For many hydraulic engineering problems the analysis of flow in open channel with a complex geometry is a fundamental prerequisite. In this research project, experimental results of longitudinal velocity profile and depth averaged velocity distribution of a compound meandering channel at the bend apex are calculated and compared with the results of this channel using ANSYS - FLUENT. To solve numerically an investigation has been carried out for studying different flow characteristics of a compound meandering channel. In this numerical study Volume of Fluid method (Two phase model) and Large Eddy Simulation (LES)turbulence model has been adopted. The former one is adopted to analyze the turbulent structure and the later one is used to predict the flow characteristics of a meandering compound channel. The sinuosity index is one for straight channel whereas for meandering channel it is greater than 1and can increase to infinity for a closed loop (where the shortest path length is zero.*

Keywords: *Large eddy simulation, Meandering Channel, Turbulent structure, Two phase model, Volume of fluid.*

I. INTRODUCTION

River is the main source of providing water supply for domestic, irrigation, industrial consumption and transportation uses. A meander, in general, is a bend in a sinuous watercourse or river. A meander is formed when the moving water in a stream erodes the outer banks and widens its valley. River channels do not remain straight for any appreciable distance. Flow separation in open channel expansion has been identified as one of the major problems encountered in many hydraulic structures. In meandering channels due to secondary effect, flow characteristics in channel bends are more complicated than that of straight channels. Due to secondary flow engery losses taken place and less energy the river deposits what it is carrying. The sinuosity index is one for straight channel whereas for meandering compound channel it is greater than 1 and can increase to infinity for a closed loop (where the shortest path length is zero) or for an infinitely- long actual depth. In straight channel River flow in the curved reach is characterized by the helical motion, or transverse circulation, due to the difference of centrifugal force between upper and lower layer of flow. The transverse circulation is associated with such featured of the curved reach including the cross –sectional shape, power expenditure, erosional and depositional pattern. An investigation is described on the distribution and magnitudes of velocity distribution arising from subcritical flows through compound meandering rectangular channels. A series of tests were conducted to determine the velocity distribution in longitudinal direction, lateral direction. The velocity contours were obtained from the experimental results of a 60^0 bend compound meandering channel and the experimental velocity contours were validated with velocity contours obtained from numerical results. The velocities were measured with surface pitot tubes adopted for application in free surface flows. The calibration for these instruments, originally developed for air flows through smooth pipes, was found to valid for direct application in smooth channels. Coles (1956) proposed a semi-empirical equation of velocity distribution, which can be applied to outer region and wall region of plate and open channel. Coleman (1981) proposed that the velocity equation for sediment-laden flow consists of two parts, as originally discussed by Coles for clear-water flow. M. Salih Kirkgoz *et al*. (1997) measured mean velocities using a Laser Doppler Anemometer (LDA) in developing and fully developed turbulent subcritical smooth open channel flows. Wilkerson *et al*. (2005) using data from three previous studies, developed two models for predicting depth-averaged velocity distributions in straight trapezoidal channels that are not wide, where the banks exert form drag on the fluid and thereby control the depth-averaged velocity distribution. Knight *et al.* (2007) used Shiono and Knight Method (SKM), which is a new approach to calculating the lateral distributions of depth-averaged velocity and boundary shear stress for flows in straight prismatic channels also accounted secondary flow effect. D. Alan Ervine, *et al*. (2000) presented a practical method to predict depth-averaged velocity and shear stress for straight and meandering overbank flows. An analytical solution to depth-integrated turbulent form of the Navier-Stokes equation was presented that included lateral shear and secondary flows in addition to bed friction. Patra and Kar (2004) reported the test results concerning the flow and velocity distribution in meandering compound river sections. It is concluded from literature review that very less work has been done regarding velocity distribution in lateral direction in rectangular compound meandering channel.. However lack of qualitative and quantitative experimental data on the lateral distribution of velocity in meandering channels is still a matter of concern to avoid may river problems like flooding. The present study aims at collecting velocity data from wide meandering channel trapezoidal at cross section. The objective of the present work is listed as:

- To study the distribution of velocity in lateral direction at channel bend apex for single flow depth.
- To validate the experimental velocity contour with three dimensional model ANSYS-FLUENT for overbank flow conditions.
- To simulate a 60 degree compound meandering channel using Large Eddy Simulation (LES) model and to derive 3 dimensional velocity contours for the same.

Various numerical models such as standard k-ε model, non-linear k-ε model, k-ω model, Algebraic Reynolds stress model(ASM), Reynolds stress model (RSM) and large eddy simulation (LES) have been developed to simulate the complex secondary structure in compound meandering channel. The standard k- ε model is an isotropic turbulence closure but fails to reproduce the secondary flows. Although nonlinear k- ε model can simulate secondary currents successfully in a compound channel, it cannot accurately capture some of the turbulence structures. ASM is economical because it uses adhoc expressions to solve Reynolds stress transport equations. But the simulated results by ASM found to be unreliable. Reynolds stress model (RSM) computes Reynolds stresses by directly solving Reynolds stress transport equation but its application to open channel is still limited due to the complexity of the model. Large eddy simulation (LES) solves spatially-averaged Navier-Stokes equation. Large eddies are directly resolves, but the eddies smaller than mesh are modelled. LES is computationally very expensive to be used for industrial application.

II. EXPERIMENTAL SETUP AND MEASUREMENTS

Experiment was conducted in a compound meandering channel having rectangular cross section built inside a concrete flume measuring 14*2*0.3 at National Institute of Technology Rourkela Hydraulic Laboratory. The width ratio of the channel is α=5.964 and the aspect ratio is δ=2.33.The length of the channel is 9.67m. The channel is made up of cement concrete. Fig.1 shows the schematic diagram of experimental setup of the open channel flow. In this paper experimentally calculated results of 1Dr value is taken for analysis with numerical analysis results of same depth and position at bend apex. Fig.2 shows the plan view of the experimental sections of a compound meandering channel

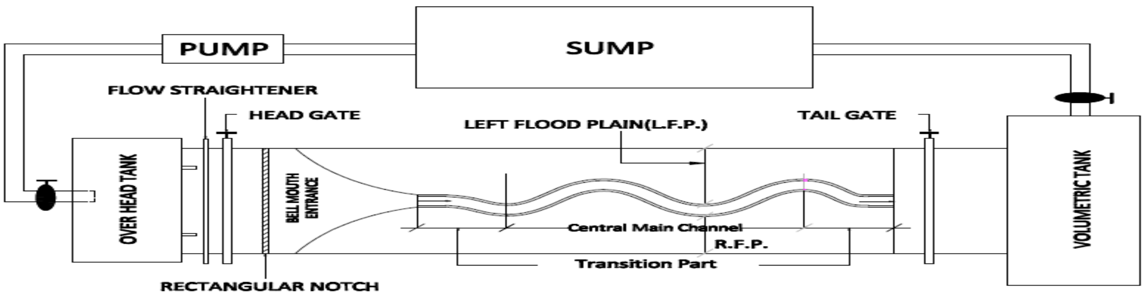


Fig.1. Plan view of experimental setup of the channel

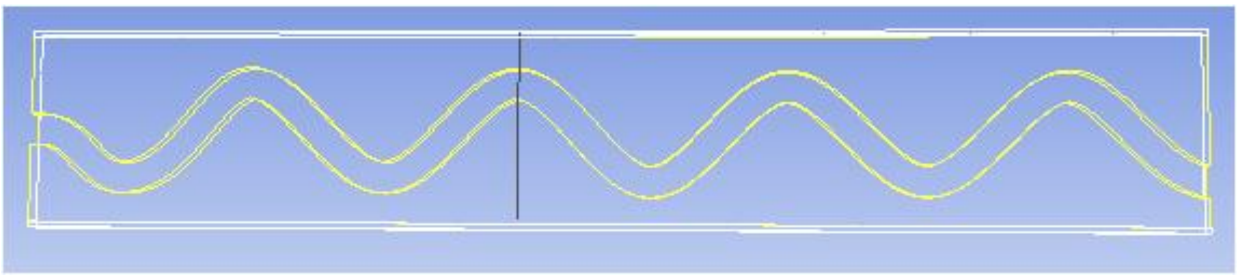


Fig.2. Plan view of the experimental sections(Bend Apex)

III. DESCRIPTION OF NUMERICAL MODEL PARAMETERS:

The fundamental basis of almost all CFD problems is the Navier–Stokes equations, which define any single-phase fluid flow. These equations can be simplified by removing terms describing viscosity to yield the Euler equations. Further simplification, by removing terms describing vorticity yields the full potential equations. Finally, for small perturbations in subsonic and supersonic flows these equations can be linearized to yield th potential equations. There is no direct solution of the equation for flow. The N-S in vector form for single phase incompressible fluid flow can be expressed as:

$$\frac{\partial}{\partial t}\left(\rho \bar{u}_i\right) + \frac{\partial(\rho \bar{u}_i \bar{u}_j)}{\partial z_i} = \frac{\partial}{\partial z_j}\left(\mu \frac{\partial \sigma_{ij}}{\partial z_j}\right) - \frac{\partial \bar{p}}{\partial z_i} - \frac{\partial \tau_{ij}}{\partial z_j}$$

SL.NO	Item Description	Value
1	Flume size	14m*2m*0.3m
2	Geometry of main channel	Meandering
3	Bank full depth of main channel	0.12m
4	Width of main channel	0.28m
5	Slope of main channel	0.00165
6	Radius of curvature at bend apex	60^0
7	Nature of surface bed	Smooth
8	Amplitude	1.130m
9	Wave length	2.230m

The algorithms adopted to solve the coupling between pressure and velocity fields in the Navier-Stokes equations in this study was pressure implicit with splitting of operators (PISO) for the numerical analysis. PISO is a non-iterative solution method to calculate the transient problem, which converges faster]. The numerical solution requires criteria for determining the convergence of the acquired solution in iteration process. The numerical solution was converged when the residuals of the discretized equation reached a value of 0.001, or when the solution did not change with further iterations. Where σij and τij are normal and shear stress components. ūi', ūj' are time averaged instantaneous velocity component along i,j directions. p = pressure, μ = co-efficient of viscosity, ρ = density.

3.1 Discretizaton Of Domain (Meshing)

The discretization of complex computational domain is critical. These kinds of domain don't coincide with the co-ordinate lines with that of a structured grid, which leads to approximation of the geometry. The only procedure to represent complex computational domain is to use a stepwise approximation. But such an approximation is also arduous and quite time consuming. Further, the stepwise approximation introduces truncation error and that can be overcome by providing very fine Cartesian mesh. Care has to be taken in order to produce a good mesh. A mesh with too few nodes could lead to a quick solution, yet not a very accurate one. However a very dense mesh of nodes will potentially waste computational time and memory. The detailed meshing of the flow domain and geometry set up of channel is shown in Fig.3 and Fig.4.

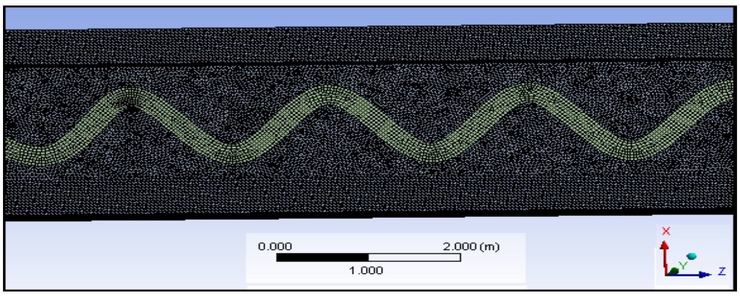


Fig.3 Plan view of meshing of compound channel

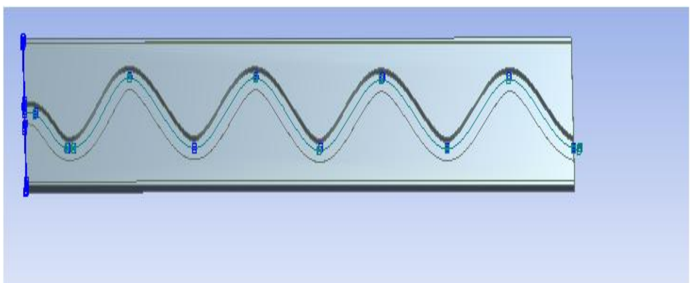


Fig.4 Plan view of geometry setup of compound channel

3.2 Boundary Condition

Wall

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.

$U = V = W = 0$

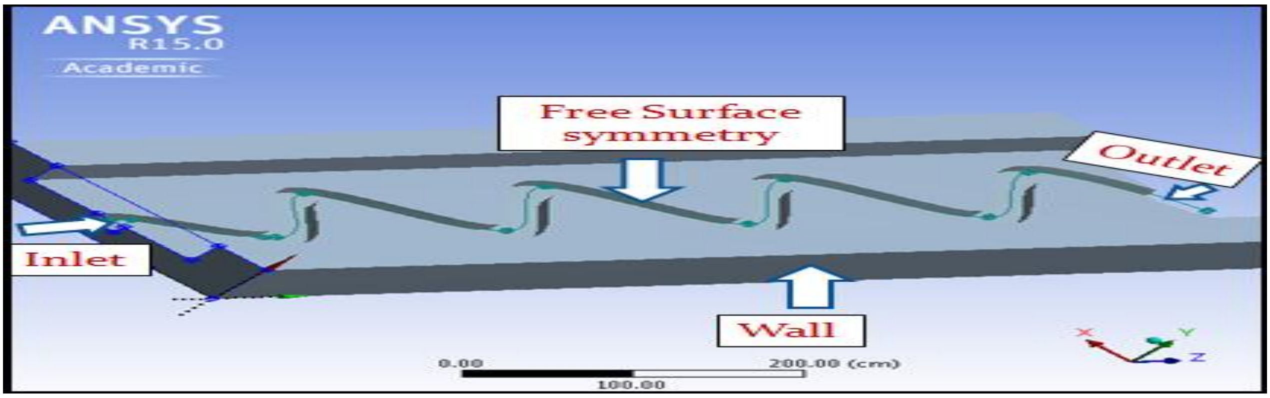
Free Surface

Here, Symmetry Boundary condition is used for the free-surface. This condition follows that, no flow of scalar flux occurs across the boundary. In applying this condition normal velocities are set to zero and values of all other properties outside the domain are equated to their values at the nearest node just inside the domain. Here the experimental bulk velocity of the flow is initially approximated as:

W = 0.328 m/s, V = 0, U= 0

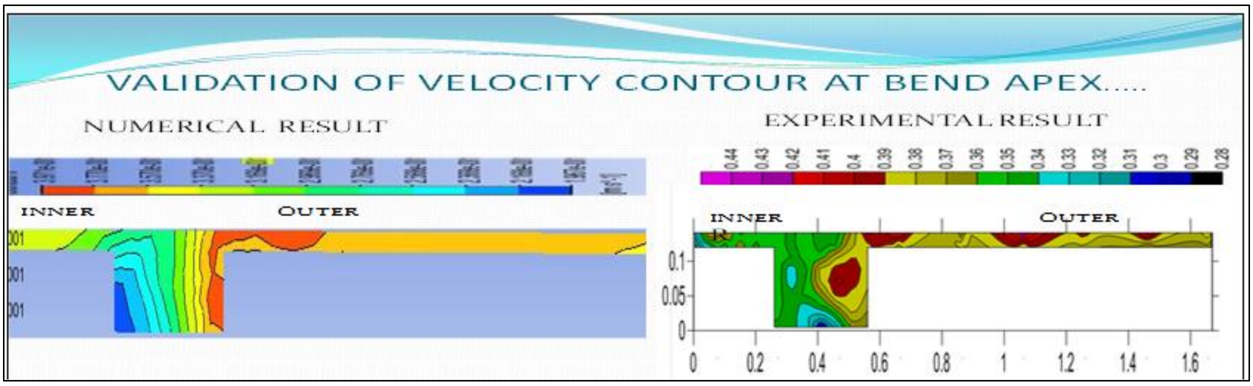
Inlet and Outlet Boundary Condition

A pressure gradient was further specified across the domain to drive the flow. In order to specify the pressure gradient the channel geometries were all created flat and the effects of gravity and channel slope implemented via a resolved gravity vector. It represents the angle between the channel slope and the horizontal, the gravity vector is resolved in x, y and z components as ($\rho g \sin\theta$ 0 $-\rho g \cos\theta$). Where θ = angle between bed surface to horizontal axis. Here, the *x* component denotes the direction responsible for flow of water along the channel and the z component is responsible for creating the hydrostatic pressure. From the simulation, *z* component of the gravity vector ($-\rho g \cos\theta$) is found to be responsible for the convergence problem of the solver.

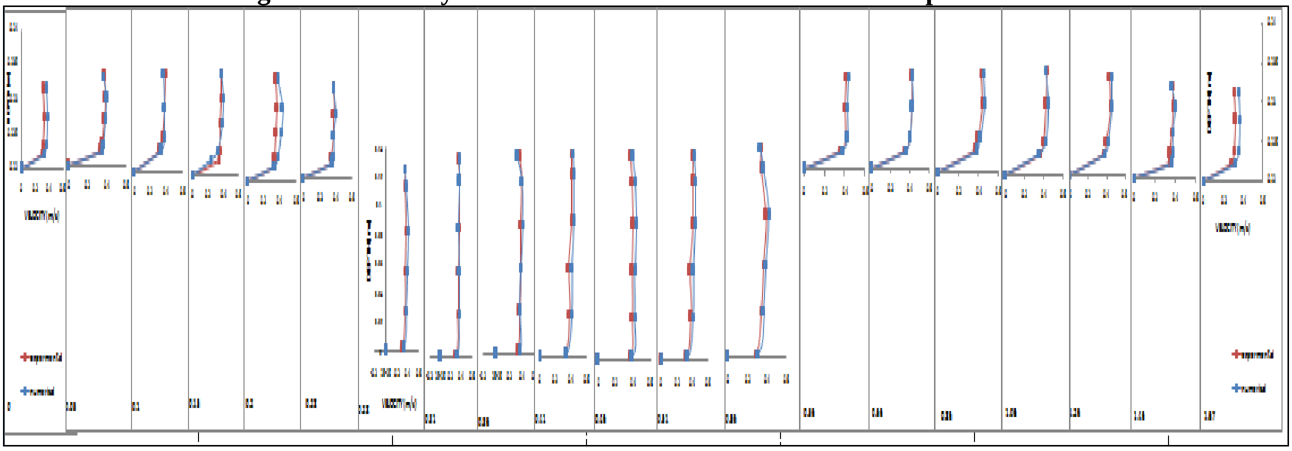


A Schematic Diagram of meandering compound channel with boundary conditions

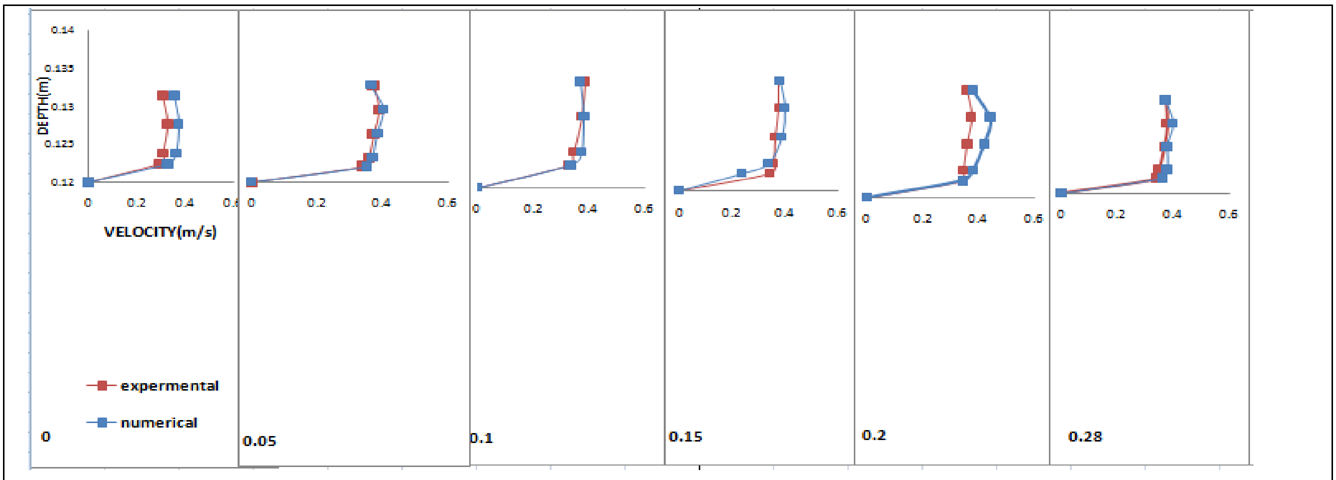
IV. NUMERICAL RESULTS



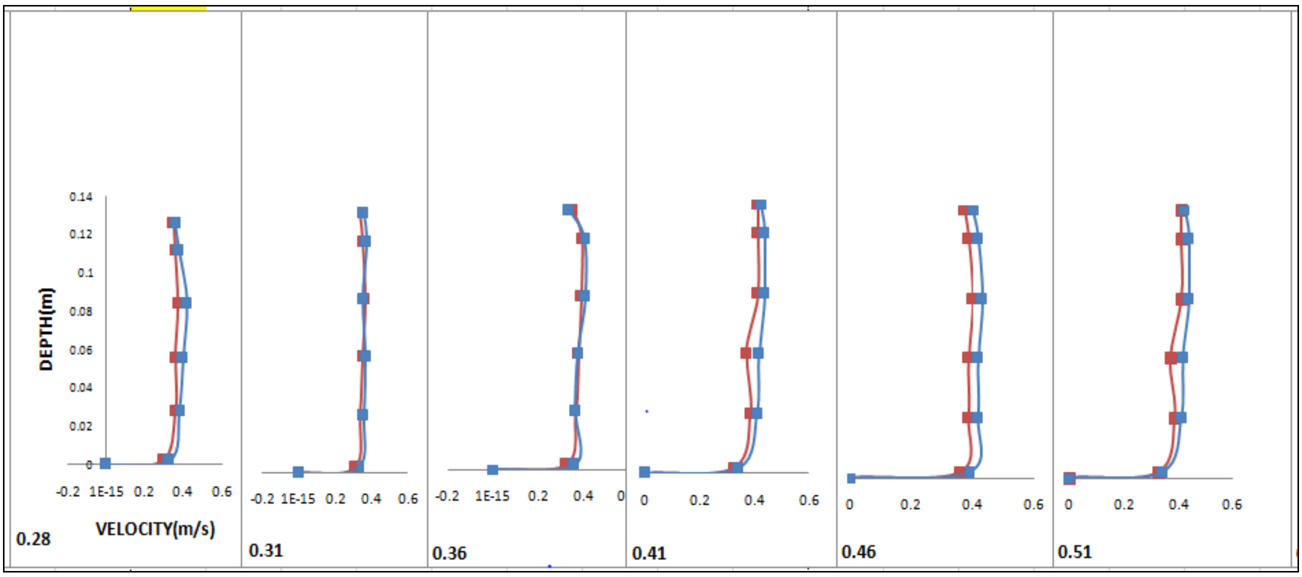
4.1Validation of Longitudinal velocity distribution in lateral direction at bend apex:



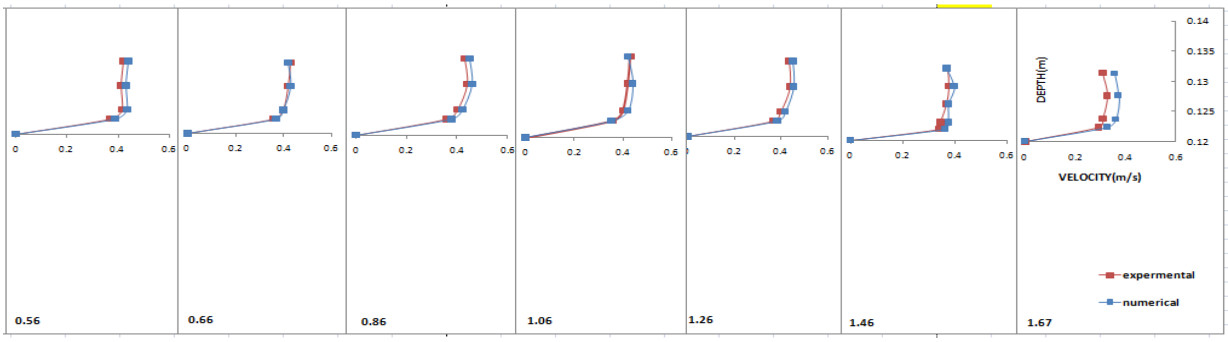
LEFT FLOOD PLAIN MAIN CHANNEL RIGHT FLOOD PLAIN



LEFT FLOOD PLAIN



MAIN CHANNEL



RIGHT FLOOD PLAIN

V. CONCLUSION

- Longitudinal velocity is recorded by pitot-tube in the experimental meandering channels. In these channels, observations are recorded at the bend apex with a direction normal to flow direction.
- The lowest velocity contour lines are found to occur at outer main channel bottom corner and its concentration increases with the increase in flow depth over the flood plain.
- The maximum value of stream wise velocity lies near the free surface and towards the inner flood plain.
- The mean velocity exists mostly in the left flood plain region.
- The occurrence of higher velocity values are more in inner flood plain region than that of outer flood plain region.

REFERENCES

[1] Ervine, D.A., et al. 2000. Two-Dimensional solution for straight and meandering overbank f lows. J. Hydr. Engrg, Vol. 126, No. 9, 0653-0669.

[2] Jia, Y., Blanckaert, K. & Wang, S.S. 2001. Numerical simulation of secondary currents in curved channels. Proc. 8th FMTM-congress, Tokyo

[3] Patra, K.C. 2004. Flow and Velocity Distribution in Meandering Compound Channels. J.Hydr. Engrg , Vol. 130, No. 5, 5-398 – 411.

[4] Kang H, Choi SU. (2006) "Reynolds stress modeling of rectangular open-channel flow. International Journal for Numerical Methods in Fluids 2006; 51:1319 – 1334.

[5] Yang K, Cao S, and LiuX. 2007. Flow resistance and its prediction methods in compound channels, Acta Mechanic Sinica; 21, pp.353-61Yen BC. (2002). Open flow resistance. J. Hydr. Eng, 1(20), pp.20-39.

[6] Orgaz, O.C. 2010. Velocity Profile and Flow Resistance Models for Developing Chute Flow. J. Hydr. Engrg , Vol. 136, No. 7, 7-447– 452.

[7] Cater, J.E., and Williams, J.R. 2008. Large eddy simulation of a long asymmetric compound open channel. Journal of Hydraulic Research. Vol. 46(4).

Thesis:

[1] Beaman F.(2010). Large Eddy Simulation of open channel flows for conveyance estimation. Ph.D thesis,University of Nottingham.

[2] Kar , S.K. (1977). "A Study of Distribution of Boundary Shear in Meander Channel with and without Floodplain and River Flood plain Interaction." PhD Thesis Presented to the IIT, Kharagpur, India.