Flow Modelling of Boundary Shear Stress Distribution for a Converging Compound Channel using ANSYS

Shiba Shankar Satapathy[1], B. Naik[2], Rahul Sahoo[3], K.K.Khatua[4]

[1,3] M. tech. Scholar, Department of Civil Engineering, N.I.T. Rourkela, India
[2] Ph. D Research Scholars, Department of Civil Engineering, N.I.T. Rourkela, India
[4]Associate Professor, Department of Civil Engineering, N.I.T. Rourkela, 769008, India
Email: shiba.628@gmail.com

Abstract

The computation of boundary shear stress distribution in a converging compound channel flow plays a significant role in various engineering problems such as modelling and designing of channels, determination of total energy losses in a channel or river, sedimentation phenomena, analysis of bed load transfer and in the study of shear stress distribution. The objective of this flow modelling for converging compound channel is to find out the shear stress distribution at the walls and on the bed of converging compound channel. Various experiments are conducted in laboratory under different discharge and relative depths. These are further analysed and compared with the results of modelling in 3-Dimensional modelling software ANSYS 15.0 whose prime purpose is to obtain contours of longitudinal, lateral and resultant body shear stress for converging compound channel of 9^0 using K-Omega model.

Keywords: *boundary shear stress, converging compound channel, relative depth, ANSYS 15*

1. Introduction

Rivers have always played an important role in the development of ancient civilization. From being used as a source of water, for obtaining food, for transport, as a defensive measure, to being the largest source of hydropower to drive machinery, Rivers have proved to be of utmost importance in our lives. In spite of this, the research growth in perceiving the hydraulic characteristics of Natural River flood plains is not satisfactory. Recent hazards and disasters clearly states that the aftermath resulting from flood can turn out to be catastrophic in nature. Therefore the flood protection methods need to be studied precisely and ahead of time in order to forecast the water levels that are the outcome of any flood discharge. Numerical analysis of compound channel flow has not been explored in the past as compared to the number of experiments that have been conducted on compound channel. Fortunately the advancement in sophisticated CFD techniques has given rise a lot more scope to the research areas in this field now. Not only CFD techniques have proved to be most economical and time saving, they also cover the aspects of flow behavior which are very difficult to determine by experimentation. A number of CFD packages such as Fluent, CFX, Star-CD are now available in market and are used for research in water flows. (Wormleaton(2005),Nguyan et al (2007), Zhang et al(2008), Shiono et al(2008)) have made a good use of these software packages for prediction of different aspects of 3D flow fields. In their work, they detected that flow characteristics in a compound channel depend on factors like topography, sinuosity, surface roughness, convergence etc. Despite such progress in CFD, alteration in flow behavior still remains an unresolved phenomenon. Since it's not practically possible to predict the flow behavior from experiments, the researchers have utilized various numerical models in order to predict the flow behavior. This problem arises due to errors that are generated in numerical modeling, such as grid generation, choice of turbulence model, discretizing scheme, specifying the initial and boundary conditions etc.

This paper highlights the flow modelling of boundary shear stress distribution for a converging compound channel which can be further used to analyse flood inundation in a river. The present work investigates effects of boundary shear stress distribution in a converging compound channel by using experimental data of a compound channel with different angles. In research the results and observations have been compared and modelled in ANSYS 15.

2. Experimental setup and procedure

Experiments were conducted in non-prismatic compound channels with varying cross section built inside a concrete flume measuring 15m×.9m×0.5m at National Institute of Technology Rourkela Hydraulic laboratory. The width ratio of the channel is $\alpha \leq 1.8$and the aspect ratio is $\delta \geq 5$. The converging angle of the channel is 9°.Converging length of the channel is 1.26m.The inside channel is made up of Perspex sheets. Water will be supplied through a Centrifugal pumps (15 hp) discharging into a RCC overhead tank. In the downstream end there will be a measuring tank followed by a sump which will feed to overhead tank through pumping thus completing recirculation path. Point velocities were taken at the grid points. The grid points in an experimental section are spaced horizontally by 5 cm and vertically by 0.2 multiplied by the depth of water at that point. Fig.1 shows the schematic diagram of experimental setup and dimensions of channel with test section respectively.Fig.2 shows the grid points in an experimental section where point velocities were recorded. Fig.3 shows different experimental sections. Table 1 shows the details of Experimental parameters for Converging Compound Channel. (Naik 2014)

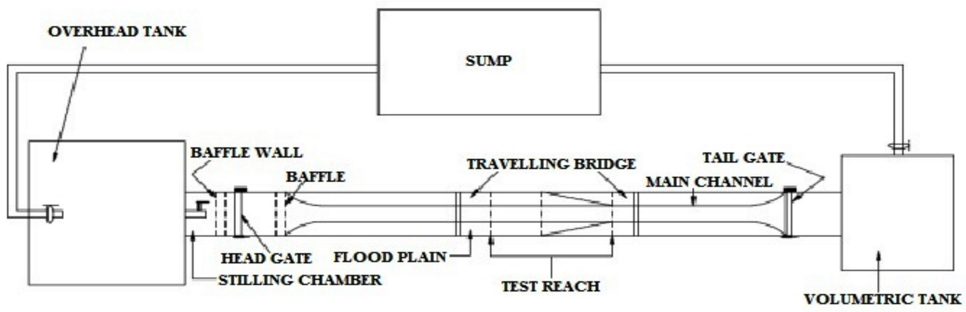


Figure1 Plan view of Experimental Setup

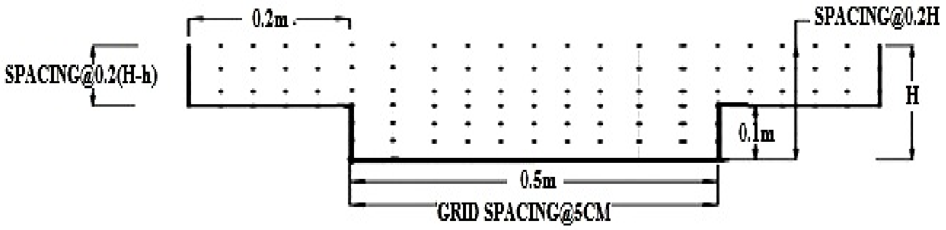


Figure 2 grid points in an experimental section where point velocities were recorded

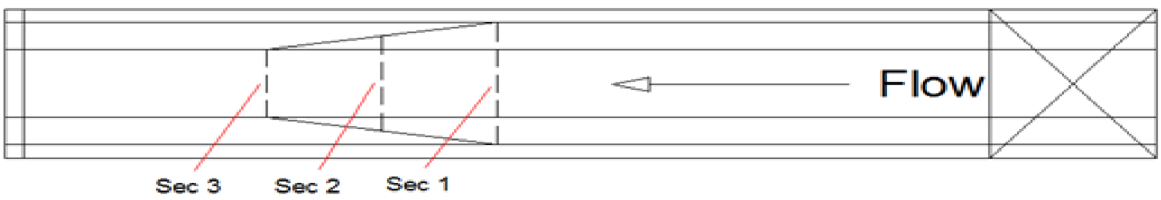


Figure 3 Experimental sections

Table1.Hydraulic parameters for the experimental channel data sets

Sl. No.	Description	Experimental Channel
1	Channel Type	Converging Compound Channel with smooth flood plain
2	Flume Size	15m*0.9m*0.5m
3	Geometry of Main Channel	Rectangular
4	Geometry of flood plain	Converging
5	Channel Width	0.5m
6	Bed Slope of Channel	0.0011
7	Main Channel Height	0.1
8	Relative Depth(Dr)(Total depth(*H)*-depth of main channel(*h)/* Total depth *(H)*)	0.3
9	Convergent angle	9^0
10	Converging length of the channel	1.26m
9	Width ratio(α) = Ratio of top width (B) to main channel width (b)	$1 \leq \alpha \leq 1.8$
10	Aspect Ratio (δ)=Ratio of main channel width(B) to main channel height	5

3. Numerical modelling

The paper is based on three-dimensional form of Navier-Stokes equations in which a Computational Fluid Dynamics simulation is used to verify the fluent model. Finite Volume Method also known as FVM is the most common method that is used in CFD modelling. It consists of both structured as well as unstructured grids. The governing equations are discretized in both space and time in free-surface modelling. In the present work, K-Omega model has been used. 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 15. 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. Fig 4 and 5 shows the geometry of converging compound channel along with its meshing of angle 9^0.

4. Methodology

The process of the numerical simulation of fluid flow generally involves four different steps and the details are given below.
(a) Problem identification
1. Defining the modelling goals
2. Identifying the domain to model
(b) Pre-Processing
1. Creating a solid model to represent the domain (Geometry Setup)
2. Design and create the mesh (grid)
(c) Solver
1. Set up the physics
•Defining the condition of flow (e.g. turbulent, laminar etc.)
•Specification of appropriate boundary condition and temporal condition.
2. Using different numerical schemes to discretize the governing equations.
3. Controlling the convergence by iterating the equation till accuracy is achieved
4. Compute Solution by Solver Setting.
•Initialization
•Solution Control
•Monitoring Solution
(d) Post processing
1. Visualizing and examining the results
2. X-Y Plots
3. Contour Draw

4.1 PREPROCESSING

In this initial step all the necessary information which defines the problem is assigned by the user. This consists of geometry, the properties of the computational grid, various models to be used, and the number of Eulerian phases, the time step and the numerical schemes.

4.1.2 Creation Geometry

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

4.1.3 Mesh generation

Second and very most important step in numerical analysis is setting up the grid associated with the construction of geometry. The Navier-Stokes Equations are non-linear partial differential equations, which consider the whole fluid domain as a continuum. In order to simplify the problem the equations are simplified as simple flows have been directly solved at very low Reynolds numbers. The simplification can be made using what is called discretization. Construction of mesh involves discretizing or subdividing the geometry into the cells or elements at which the variables will be computed numerically.

4.1.4 Solver setting

The numerical scheme that CFD codes adopt is the finite volume method. The differential transport equations are integrated over each computational cell, and the Gauss and Leibnitz theorems are applied in this method. This consists of various models used for analysis, the initial and boundary conditions, the number of Eulerian phases, the properties of the materials, the physical and chemical phenomena involved. At last, the set of algebraic equations is solved by iteration process and the cell-center values of the flow variables are calculated.

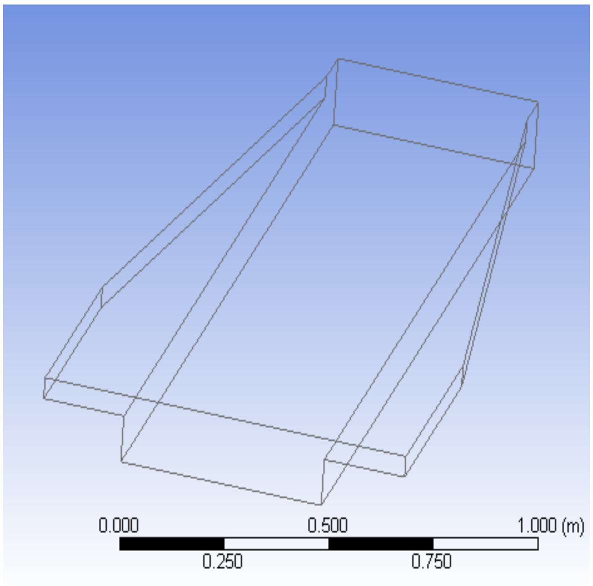


Figure 4 Geometry of Converging Compound Channel

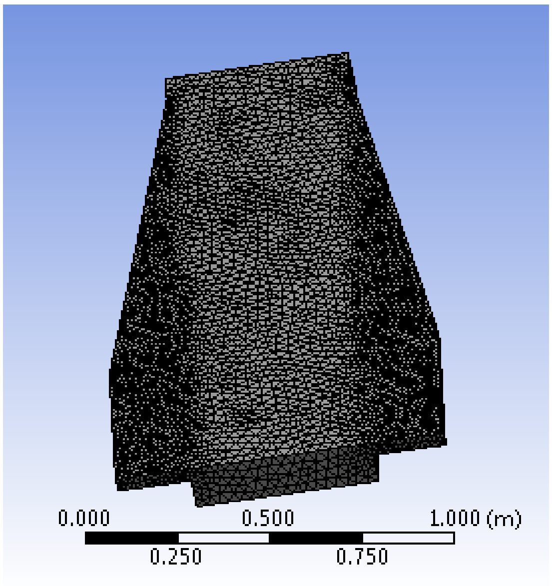


Figure 5 Meshing of converging Compound Channel

5. Results and discussion

Point velocities at grid points of the three different cross sections are collected and the velocity contour of each cross section is plotted using software called Surfer. The CFD model of similar specifications as experimental channel was simulated and again the velocity contour is extracted. Velocity contours from both experimental and numerical analysis are compared. It was found out that the point velocities are close to each other with a standard deviation of 10-12%. Figures 6, 7,8,9,10,11 shows the transverse velocity contours of sec-1, sec-2, sec-3 respectively for relative depths 0.3. Similarly figure 12, 13, 14 shows the both numerical and experimental boundary shear stress of sec-1, sec-2 and sec-3 respectively. From this figures we concluded that boundary shear increases from sec-1 to sec-3 gradually.

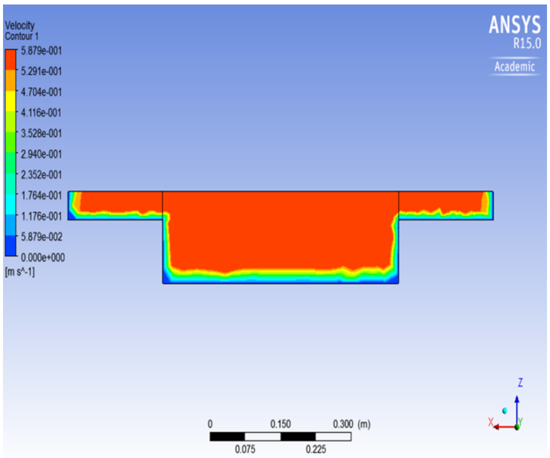


Figure 6 Transverse Velocity Contour of Sec 1 by Numerical Analysis

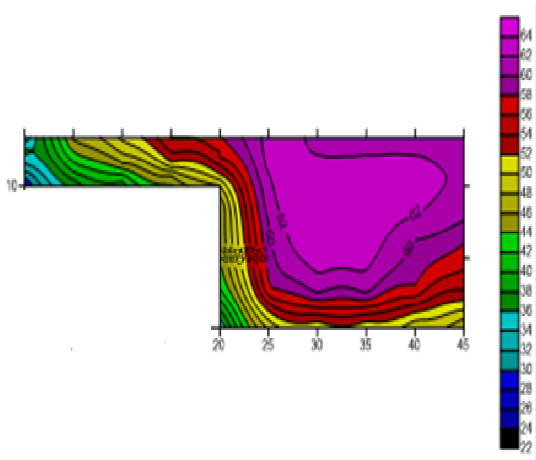
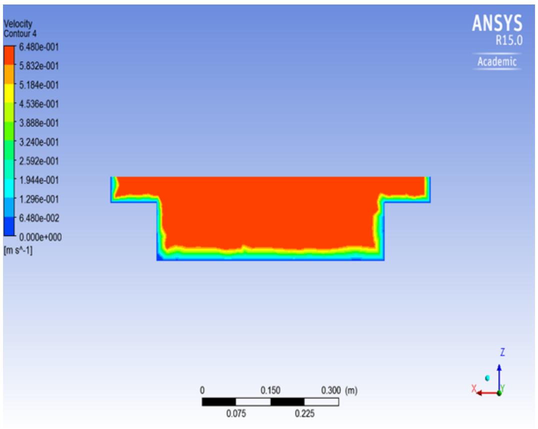


Figure 7 Transverse Velocity Contour of Sec 1 from Experiment



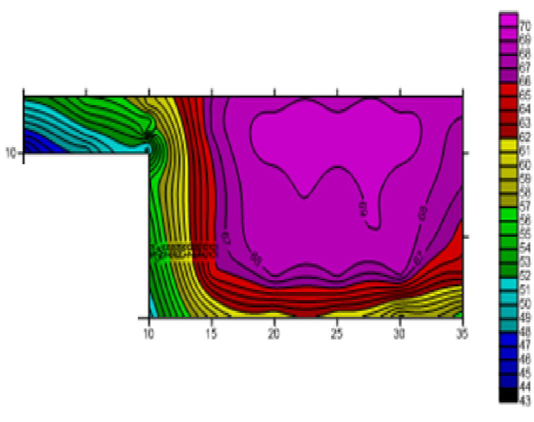
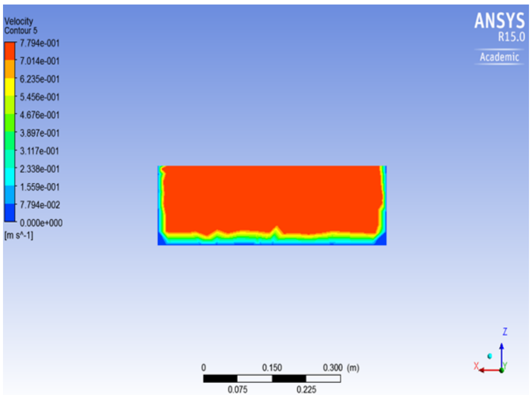


Figure 8 Transverse Velocity Contour of Sec 2 by Numerical Analysis

Figure 9 Transverse Velocity Contour of Sec 2 from Experiment



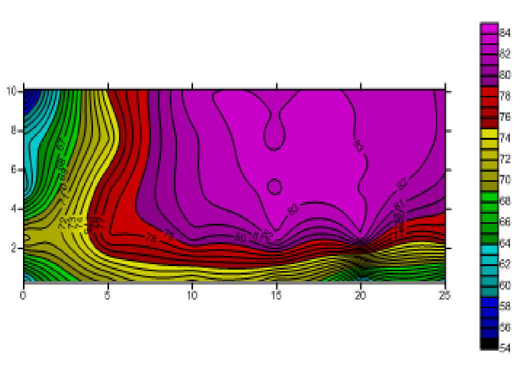


Figure 10 Transverse Velocity Contours of Sec 3 by Numerical Analysis

Figure 11 Transverse Velocity Contours of Sec 3 from Experiment

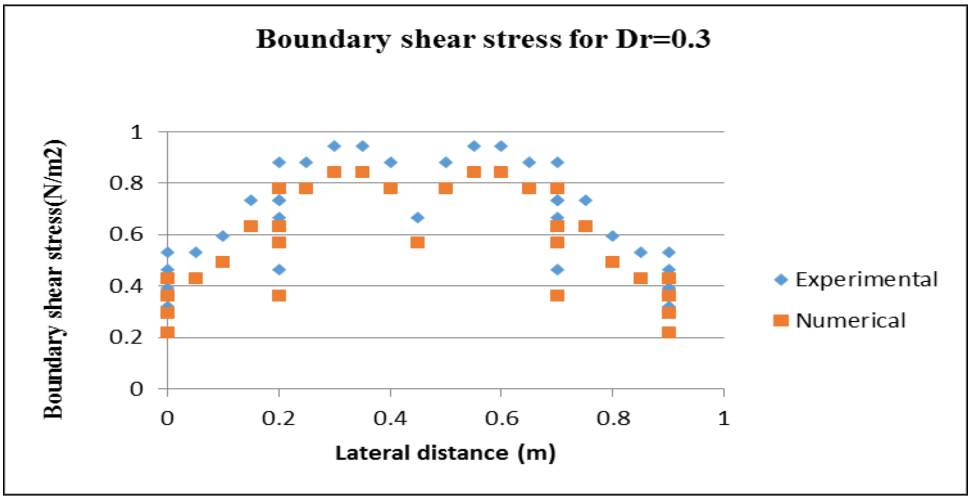


Figure 12 Boundary shear stress of Sec 1

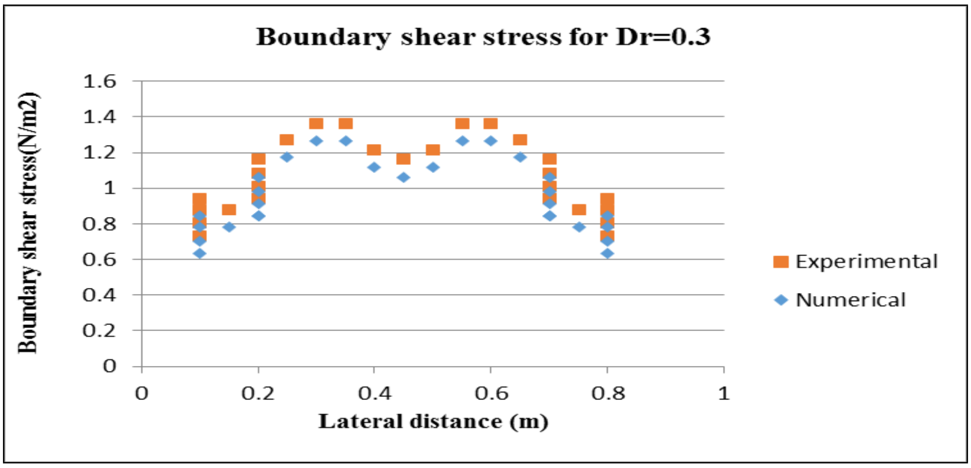


Figure 13 Boundary shear stress of Sec 2

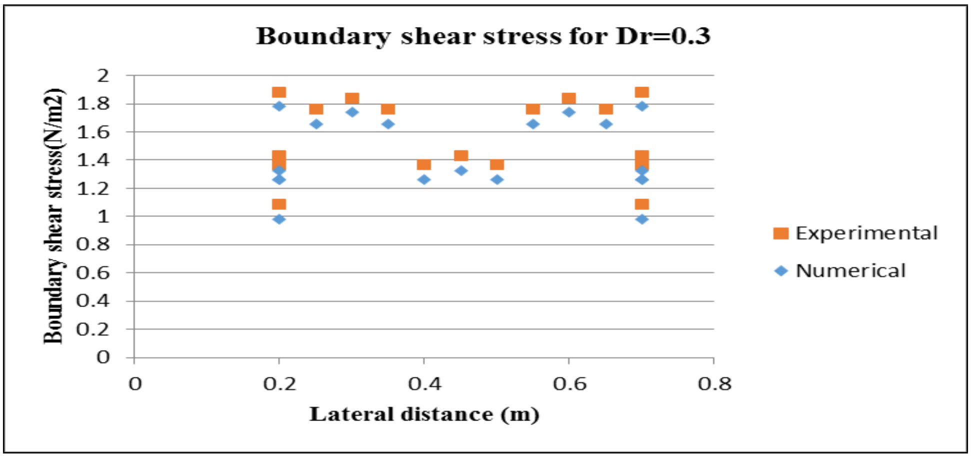


Figure 14 Boundary shear stress of Sec 3

6. Conclusions

In the present study, the experimental and numerical modelling of the flow pattern at a non-prismatic compound channel with a converging flood plain has been carried out. On the basis of the investigations concerning boundary shear stress distribution along the channel bed for relative depth 0.3 in a non-prismatic converging compound channel having 9° converging angle, the point to point observations were drawn and the following conclusions were made:

1. Three dimensional modelling of converging compound have been successfully verified with a mesh refinement studies and validated with experiments.

2. The experimental results that were observed and related to low velocity contours are then compared with the corresponding values obtained from Numerical Analysis for 3 different sections i.e. Sec-1, Sec-2, and Sec- 3 for the converging compound channel of relative depth 0.3. A good agreement between the results of the CFD analysis and Experimental results was seen. The velocity increases along the channel, as the cross-section area decreases in both the cases.

3. From the boundary shear stress distribution, it can be clearly seen that the numerical analysis provides good results and it is slightly underestimating the data when it is compared with the experimental result with a positive error of more or less than 5%.

7. Acknowledgements

The author wish to acknowledge thankfully the support from the Institute and the UGC UKIERI Research project (ref no UGC-2013 14/017) by the second authors for carrying out the research work in the Hydraulics Laboratory at National Institute of Technology, Rourkela is thankfully acknowledged.

8. References

A.Ramamurthy, S.Han, P.Biron, (2013). Three Dimensional Simulation Parameters for 90Â° Open Channel Bend Flows, *Journal of Computing in Civil Engineering*, 27(3), 282-291.

B. Naik., K.K.Khatua, and Kamel. Miri, (2014) Energy loss along the non-prismatic reach of a compound channel using ANN, *River **flow 2014***,International conference on Fluvial Hydraulics (Sept 3-5), Lausanne, Switzerland

B.K.Gandhi, H.K.Verma, B.Abraham, (2010). Investigation of Flow Profile in Open Channels using CFD",*IGHEM*, 1 -12.

Bousmar, D., Wilkin, N., Jacquemart, J.H., and Zech, Y. (2004) .Overbank flow in symmetrically narrowing floodplains. *J. Hydraul. Eng., ASCE*, 130(4), 305-312

D.W.Knight, K.Shiono, (1990).Turbulence measurements in a shear layer region of a compound channel, *Journal of Hydraulic Research*, 28(2), 175-196.

Nguyan, V.TH. Nestmann, F., and Scheuerlein, H., (2007). Three Dimensional Computation of Turbulent Flows in Meandering Channels and Rivers, *IAHR Journal of Hydraulic Research*, Volume 45, No. 5, pp. 595-609

O. Seyedashraf, Ali Akbar, M.K. Shahidi, (2012). "Numerical Study of Channel Convergence on Flow Pattern in 90 Degree Bends, *9th International Congress on Civil Engineering*, Isfahan University of Technology (IUT), Isfahan, Iran, 1 -8.

Proust, S., Rivière, N., Bousmar, D., Paquier, A., and Zech, Y. (2006) .Flow in compound channel with abrupt floodplain contraction. *J. Hydraul. Eng.,* 132(9), 958-970

Shiono, K., Spooner, J., Chan, T., Rameshwaran, J., and Chandler, J., (2008). Flow Characteristics in Meandering Channels with Non-Mobile and Mobile Beds for Overbank Flows, *IAHR Journal of Hydraulic Research*, Volume 46, No. 1,pp. 595-609

T. Bodnar, Prihoda, (2008).Numerical simulation of turbulent free-surface flow in curved channel Flow, turbulence and combustion, 76(4), 429-442.

Zhang, M.L., and Shen, Y.M., (2008). Three Dimensional Simulation of Meandering River Based on 3-D RNGTurbulence Model", *Journal of Hydrodynamics*, Volume 20, No. 4, pp. 448-455